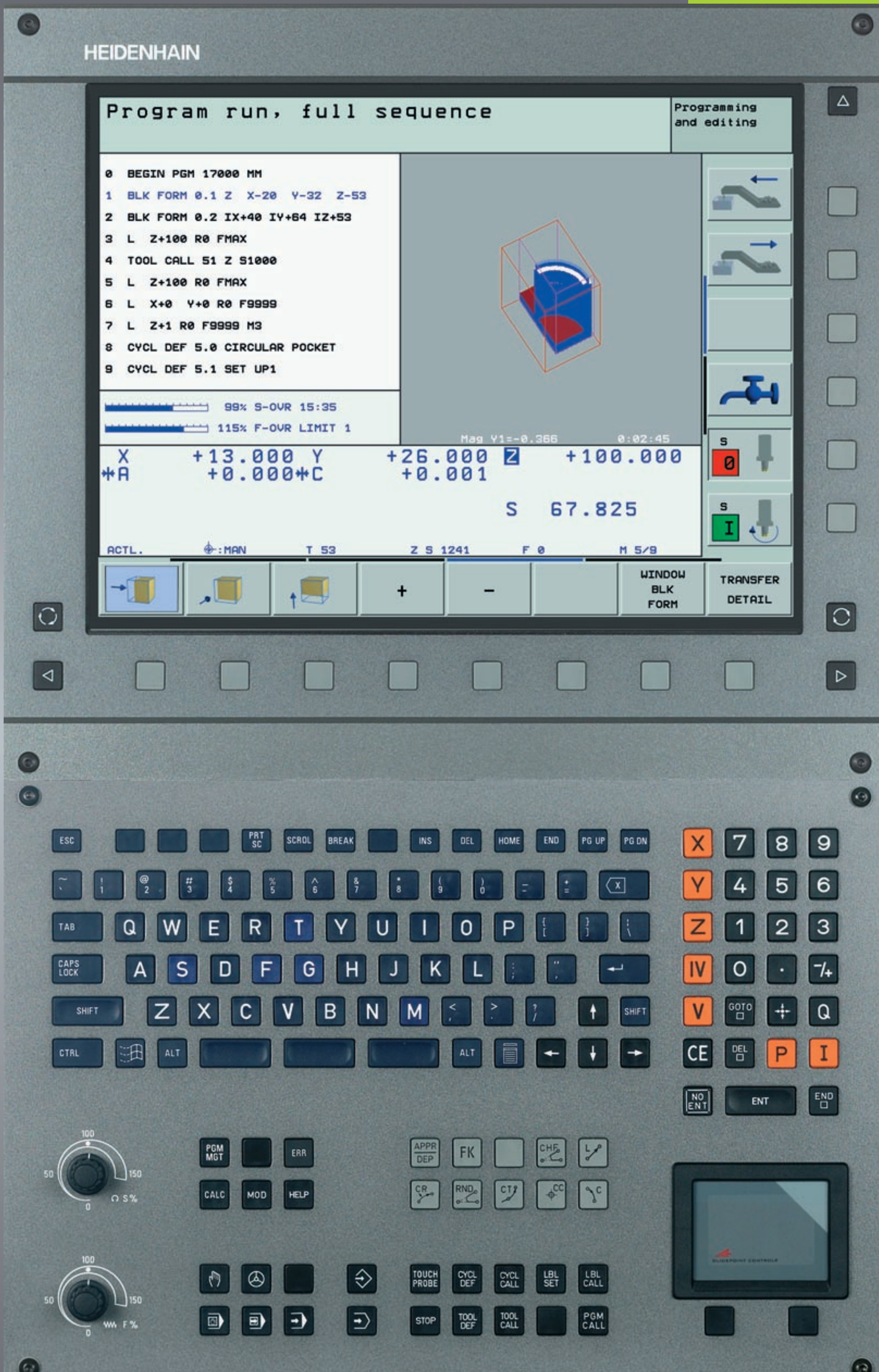


HEIDENHAIN

iTNC 530

NC Software
340 420-xx

User's Manual
ISO Programming

English (en)
7/2003

Controls on the visual display unit

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

Typewriter keyboard for entering letters and symbols

- Q W E R T Y — File names, Comments
- G F S T M — ISO programs

Machine operating modes

- MANUAL OPERATION
- ELECTRONIC HANDWHEEL
- POSITIONING WITH MDI
- PROGRAM RUN, SINGLE BLOCK
- PROGRAM RUN, FULL SEQUENCE

Programming modes

- PROGRAMMING AND EDITING
- TEST RUN

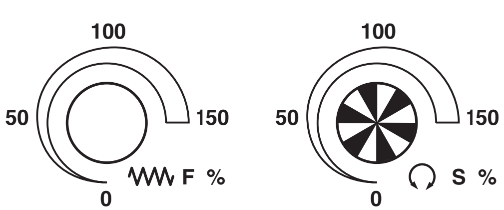
Program/file management, TNC functions

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

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

- Move highlight
- GOTO — Go directly to blocks, cycles and parameter functions

Override control knobs for feed rate/spindle speed



Programming path movements

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

Tool functions

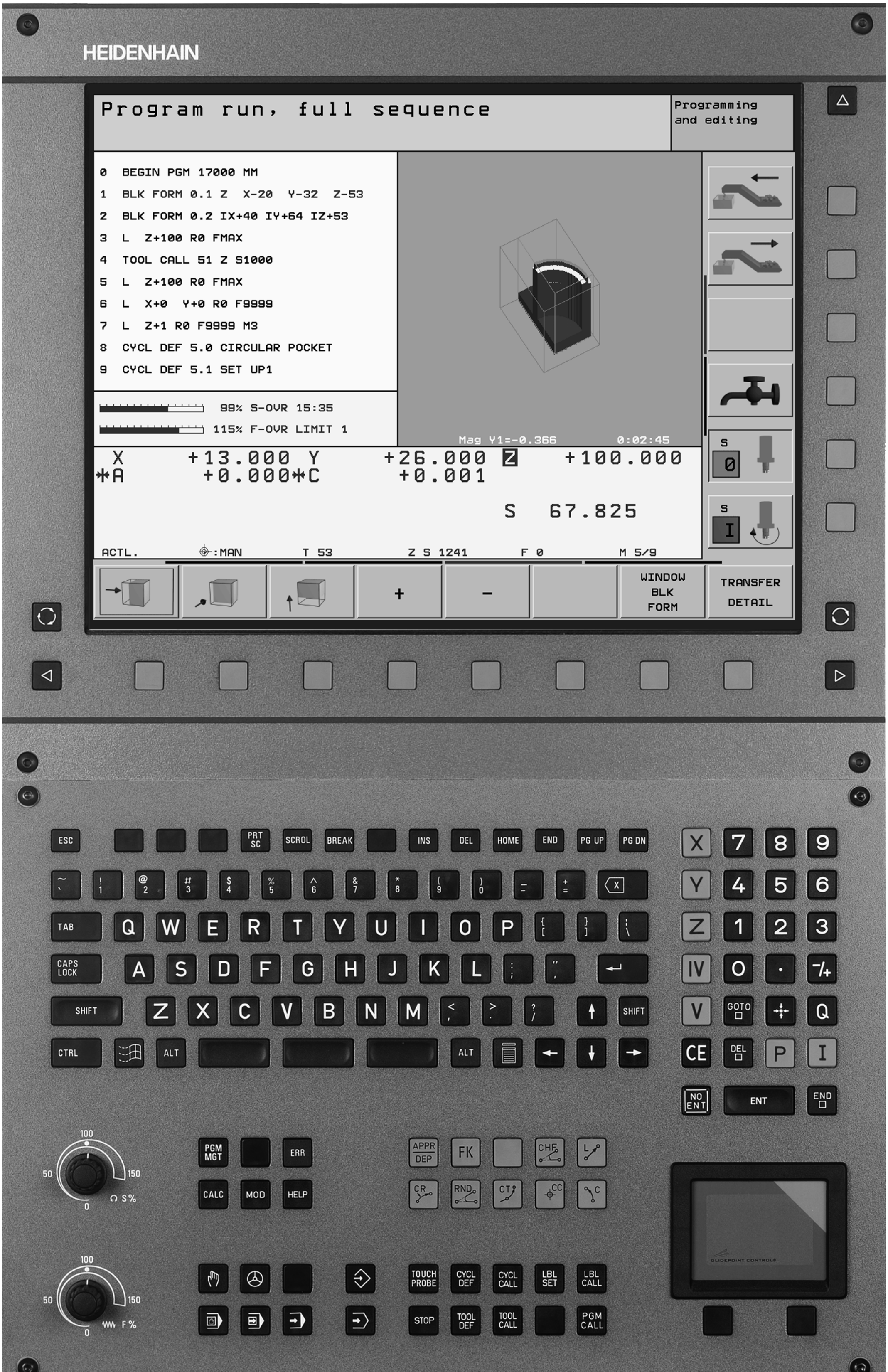
- TOOL DEF, TOOL CALL — Enter and call tool length and radius

Cycles, subprograms and program section repeats

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

Coordinate axes and numbers: Entering and editing

- X . . . V — Select coordinate axes or enter them into the 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 clear TNC error message
- DEL — Abort dialog, delete program section


HEIDENHAIN
Program run, full sequence
Programming and editing
0 BEGIN PGM 17000 MM
1 BLK FORM 0.1 Z X-20 Y-32 Z-53
2 BLK FORM 0.2 IX+40 IY+64 IZ+53
3 L Z+100 R0 FMAX
4 TOOL CALL 51 Z S1000
5 L Z+100 R0 FMAX
6 L X+0 Y+0 R0 F9999
7 L Z+1 R0 F9999 M3
8 CYCL DEF 5.0 CIRCULAR POCKET
9 CYCL DEF 5.1 SET UP1
99% S-OVR 15:35
115% F-OVR LIMIT 1
Mag Y1=-0.366
0:02:45
X +13.000 Y +26.000 Z +100.000
*A +0.000 *C +0.001
S 67.825
ACTL.
:MAN
T 53
Z S 1241
F 0
M 5/9
WINDOW BLK FORM
TRANSFER DETAIL
ESC
PRT SC
SCROL
BREAK
INS
DEL
HOME
END
PG UP
PG DN
TAB
CAPS LOCK
SHIFT
CTRL
ALT
CE
GOTO
NO ENT
ENT
END
PGM MGT
ERR
CALC
MOD
HELP
APPR DEP
FK
CHF
CR
RND
CT
CC
TOUCH PROBE
CYCL DEF
CYCL CALL
LBL SET
LBL CALL
STOP
TOOL DEF
TOOL CALL
PGM CALL
S %
F %

TNC Model, Software and Features

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

TNC model	NC software number
iTNC 530	340 420-09
iTNC 530	340 421-09

The export versions of the TNC 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
- Tool measurement with the TT 130
- Rigid tapping
- Returning to the contour after an interruption

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

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

Touch Probe Cycles User's Manual:

All of the touch probe functions are described in a separate manual. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID number: 369 280-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.

New features of the NC software 340 420-xx

- Connecting the TNC to **Windows networks** via Ethernet (see "Network settings specific to the device" on page 449)
- **Automatic cutting data calculation** in ISO programs (see "Working with Cutting Data Tables" on page 147)
- Definition of overlapping contours with **contour formula** (see "SL Cycles with Contour Formula" on page 331)
- **Structuring** ISO programs (see "Structuring Programs" on page 104)
- **Find/Replace** any text (see "The TNC search function" on page 100)
- Changing the position of the current block on the screen (see "Editing a program" on page 96)
- New Q parameter functions: **Check sign** and **Calculate modulo value** when entering formulas (see "Entering Formulas Directly" on page 400)

Changed features of the NC software 340 420-xx

- Cycle G62 Tolerance has been expanded so that different filter settings can be selected for High Speed Cutting (see "TOLERANCE (Cycle G62)" on page 369).
- In Cycle G210 (Slot with reciprocating plunge), the approach behavior for finishing has been changed (see "SLOT with reciprocating plunge-cut (Cycle G210)" on page 285).
- The number of contour elements permitted in SL Cycles, Group II, has been increased from approx. 256 to approx. 1024 (see "SL Cycles Group II" on page 306).
- ISO programs are now programmed in conversational mode (see "Creating and Writing Programs" on page 91).
- The transfer of the current tool position coordinates into the program has been improved (see "Actual position capture" on page 95).
- The transfer of the value that is calculated by using the on-screen pocket calculator into the program has been modified (see "Integrated Pocket Calculator" on page 110).
- The PGM CALL key can now be used for programming program calls (see "Calling any program as a subprogram" on page 375).
- Detail magnification is now also possible in plan view (see "Magnifying details" on page 420).
- When program sections are copied, the copied block remains highlighted after having been inserted (see "Marking, copying, deleting and inserting program sections" on page 98).

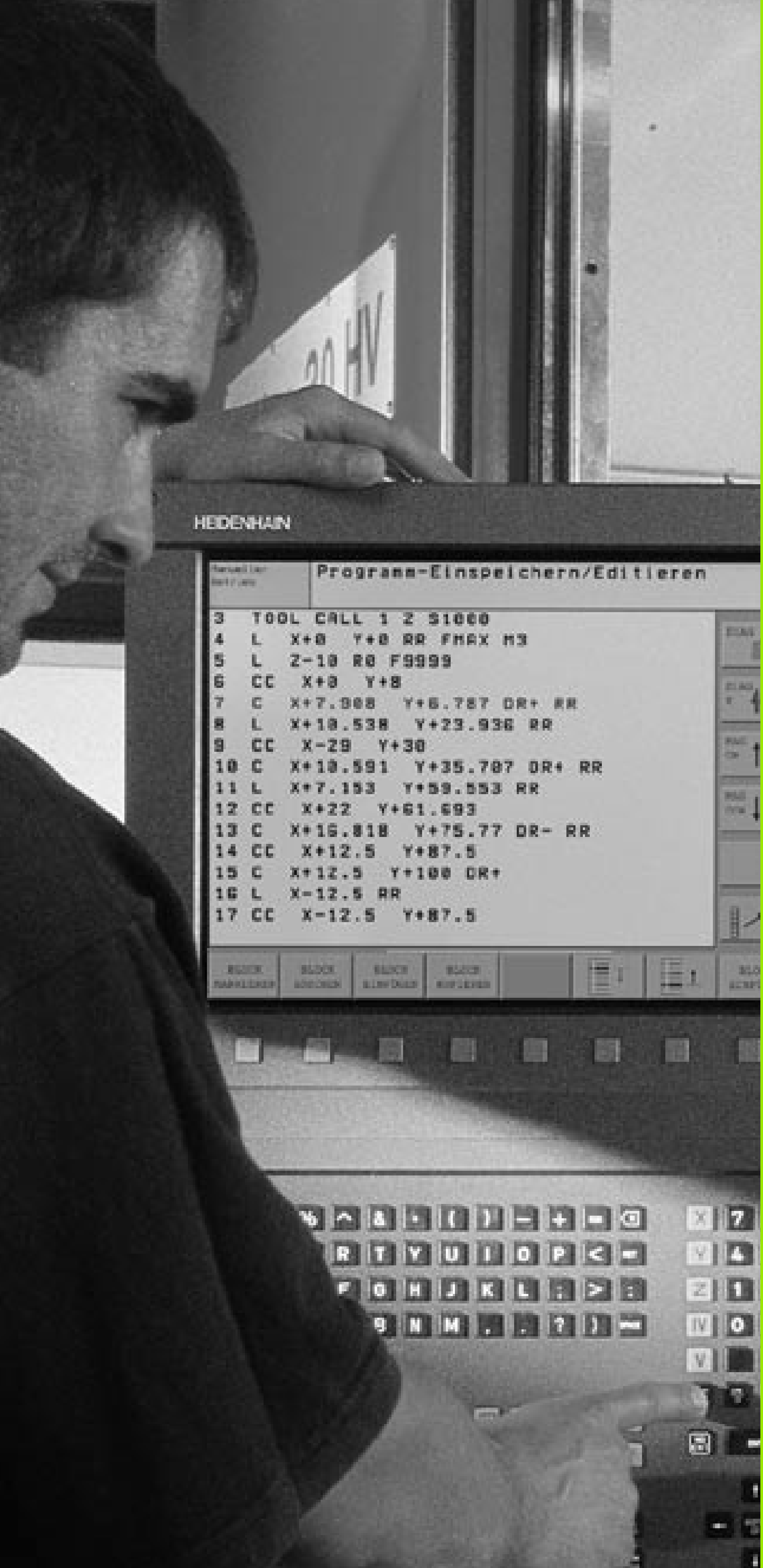
New/changed descriptions in this manual

- Example of Cycle G128 Slot Milling on Cylinder Surface added (see “Example: Cylinder surface with Cycle G128” on page 329).
- Meaning of software numbers after the MOD functions have been selected (see “Software Numbers and Option Numbers” on page 440).

Contents

1 Introduction 29

6 Programming: Programming Contours 155



1

Introduction

1.1 The iTNC 530

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 iTNC 530 can control up to 9 axes. You can also change the angular position of the spindle under program control.

An integrated hard disk provides storage for as many programs as you like, even if they were created off-line. For quick calculations you can call up the on-screen pocket calculator at any time.

Keyboard and screen layout are clearly arranged in such a 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 does 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 also enter and test one program while the control is running another.

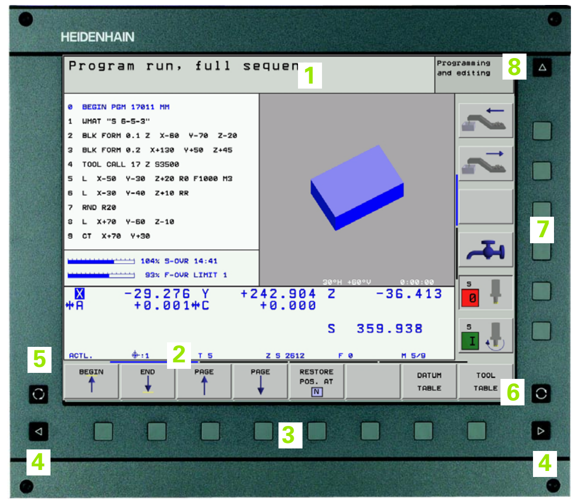
Compatibility

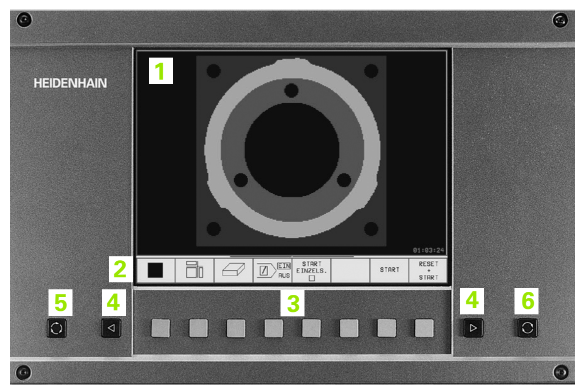
The TNC can run all part programs that were written on HEIDENHAIN controls TNC 150 B and later.

1.2 Visual Display Unit and Keyboard

Visual display unit

The TNC is available with either a BF 150 color TFT flat-panel display or the BF 120 color TFT flat-panel display. The figure at top right shows the keys and controls on the BF 150, and the figure at center right shows those of the BF 120.





1 Header

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

2 Soft keys

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

3 Soft-key selection keys

4 Switches the soft-key rows

5 Sets the screen layout

6 Shift key for switchover between machining and programming modes

7 Soft-key selection keys for machine tool builders

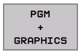
8 Switches soft-key rows for machine tool builders

Screen layout

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

To change the screen layout:

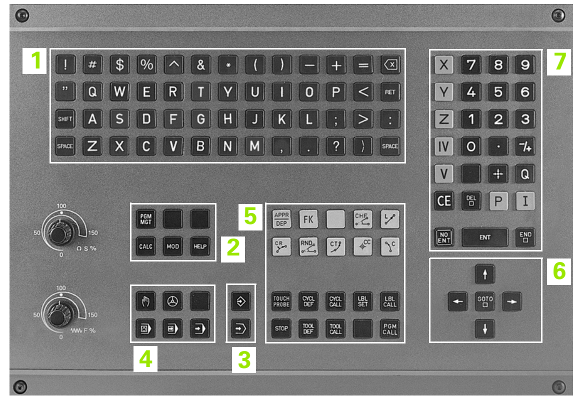
Press the SPLIT SCREEN key: The soft-key row shows the available layout options (see "Modes of Operation," page 34).



Select the desired screen layout.

Keyboard

The TNC is available either with the TE 420 or TE 530 keyboard. The figure at upper right shows the operating elements of the TE 420 keyboard; the figure at center right shows the operating elements of the TE 530 keyboard:



1 Alphabetic keyboard for entering texts and file names, and for ISO programming.

  Dual-processor version: Additional keys for Windows operation

2
- File management
- Pocket calculator
- MOD function
- HELP function

3 Programming modes

4 Machine operating modes

5 Initiation of programming dialog

6 Arrow keys and GOTO jump command

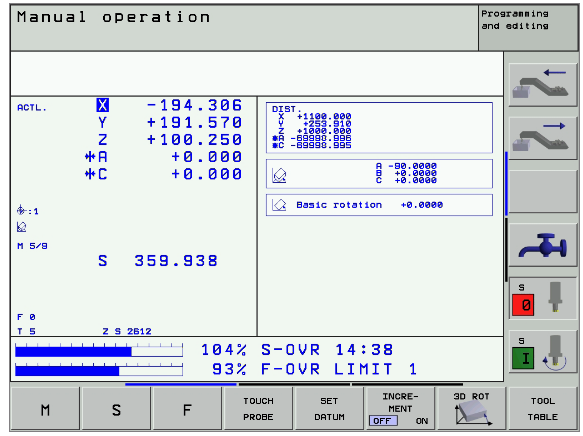
7 Numerical input and axis selection

8 Mouse pad: Only for operating the dual-processor version

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

Manual Operation and Electronic Handwheel



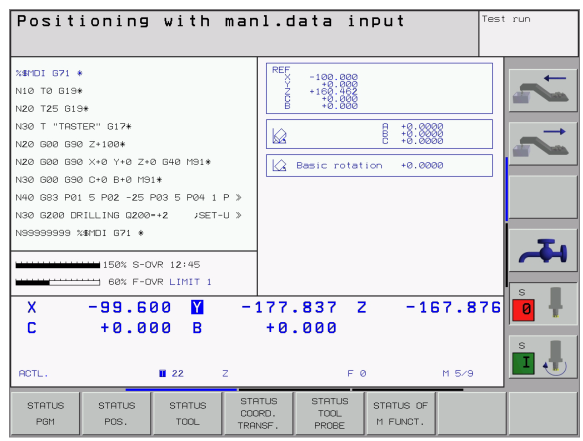
The Manual Operation mode is required for setting up the machine tool. In this operating mode you can position the machine axes manually or by increments, set the datums, and tilt the working plane.

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 described previously)

Screen windows	Soft key
Positions	POSITION
Left: positions, right: status display	POSITION + STATUS

Positioning with Manual Data Input (MDI)

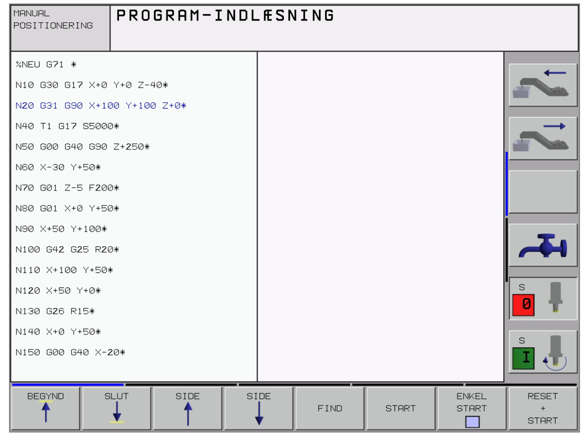


This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program blocks, right: status display	PGM + STATUS

Programming and editing

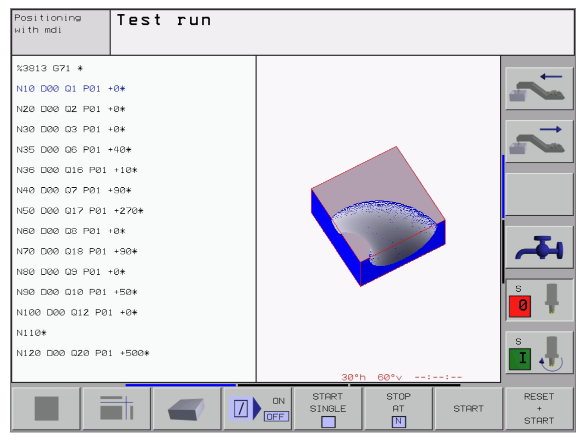


In this mode of operation you can write your part programs. The various cycles and Q-parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the individual steps.

Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program, right: programming graphics	PGM + GRAPHICS
Left: program blocks, right: program structure	PGM + SECTS

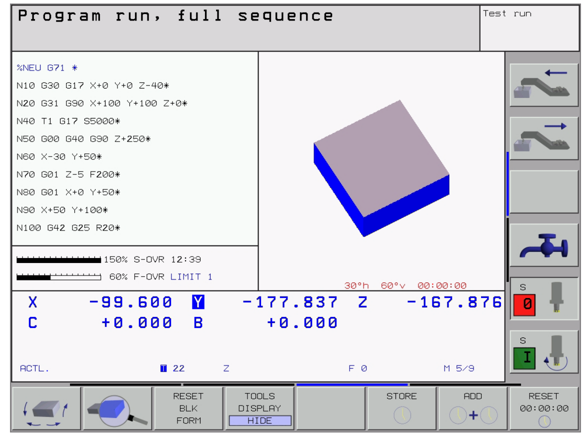
Test Run



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

Soft keys for selecting the screen layout: see "Program Run, Full Sequence and Program Run, Single Block," page 36.

Program Run, Full Sequence and Program Run, Single Block



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

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

Soft keys for selecting the screen layout

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

Soft keys for selecting the screen layout for pallet tables

Screen windows	Soft key
Pallet table	PALLET
Left: program, right: pallet table	PGM + PALLET
Left: pallet table, right: status	PALLET + STATUS
Left: pallet table, right: graphics	PALLET + GRAPHICS

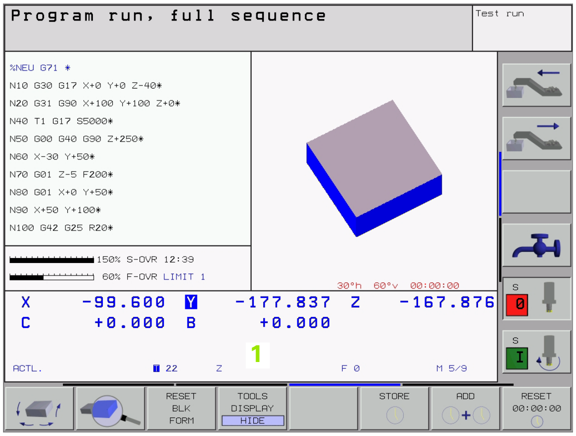
1.4 Status Displays

"General" status display

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

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

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



Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position
XYZ	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
FSM	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
	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 for the Programming and Editing mode of operation.

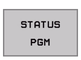
To switch on the additional status display:

Call the soft-key row for screen layout.

Select the layout option for the additional status display.

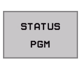
To select an additional status display:

Shift the soft-key rows until the STATUS soft keys appear.



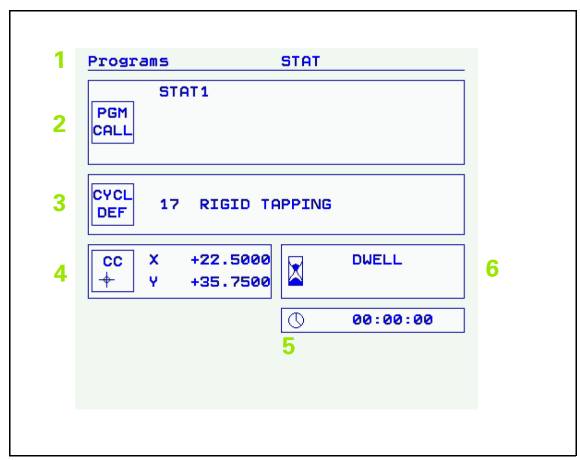
Select the desired additional status display, e.g. general program information.

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



General program information

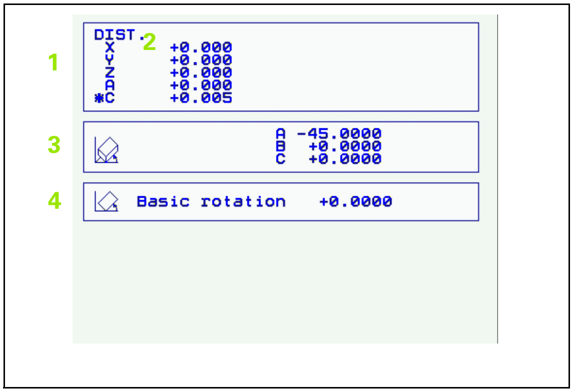
1. Name of main program
2. Active programs
3. Active machining cycle
4. Circle center CC (pole)
5. Operating time
6. Dwell time counter



STATUS POS.

Positions and coordinates

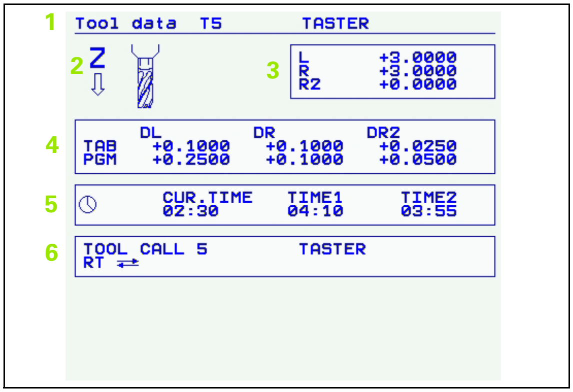
1 Position display
2 Type of position display, e.g. actual position
3 Tilt angle of the working plane
4 Angle of a basic rotation



STATUS TOOL

Information on tools

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

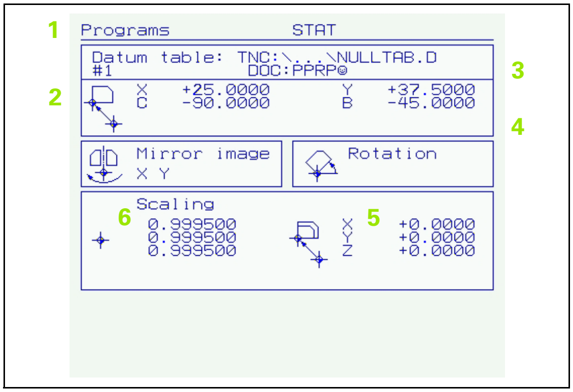


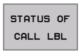
STATUS COORD. TRANSF.

Coordinate transformations

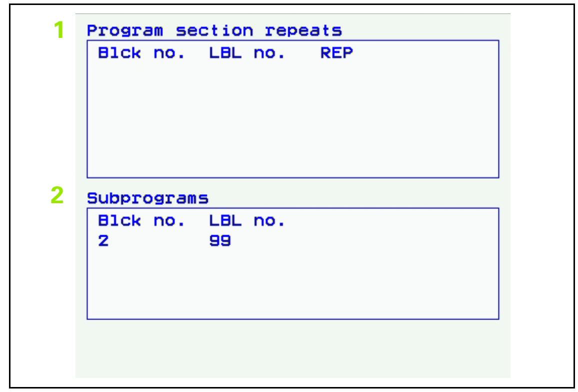
1 Name of main program
2 Active datum shift (Cycle 7)
3 Active rotation angle (Cycle 10)
4 Mirrored axes (Cycle 8)
5 Active scaling factor(s) (Cycles 11 / 26)
6 Scaling datum

See “Coordinate Transformation Cycles” on page 348.



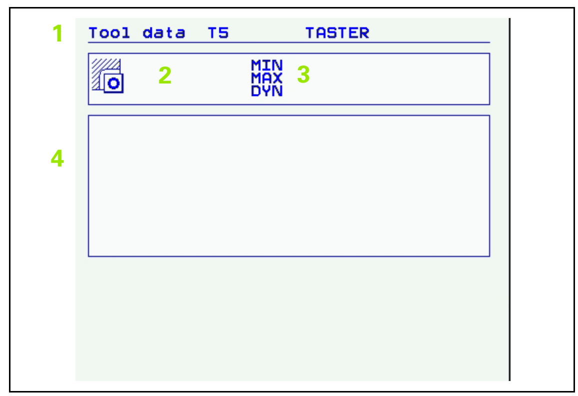


Program section repeats/subprograms



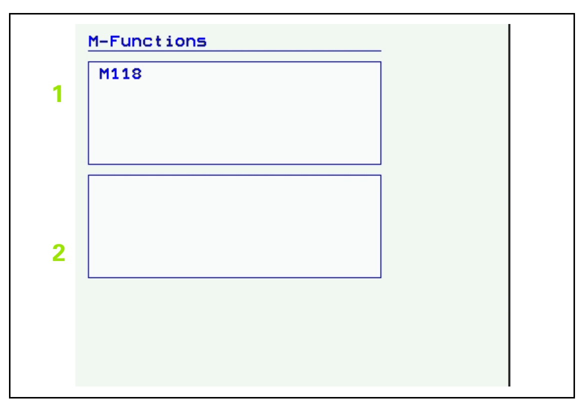
1 Active program section repeats with block number, label number, and number of programmed repeats/repeats yet to be run
2 Active subprogram numbers with block number in which the subprogram was called and the label number that was called

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

Active miscellaneous functions M



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
- Measure and inspect tools

All of the touch probe functions are described in a separate manual. Please contact HEIDENHAIN if you require a copy of this User’s Manual. Id. Nr.: 369 280-xx.

TS 220, TS 630 and TS 632 touch trigger probes

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

The TS 630 and TS 632 feature infrared transmission of the triggering signal to the TNC. This makes them highly convenient for use on machines with automatic tool changers.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear-resistant 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.

TT 130 tool touch probe for tool measurement

The TT 130 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. The TT 130 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 (see figure at center right).

2

Manual Operation and Setup

2.1 Switch-On, Switch-Off

Switch-on

Switch-on and Traversing the Reference Points can vary depending on the machine tool. Refer to your machine manual.

Switch on the power supply for control and machine. The TNC 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 external dc voltage. The TNC checks the functioning of the EMERGENCY STOP circuit.

MANUAL OPERATION
TRAVERSE REFERENCE POINTS

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

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

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

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

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 must be active in the Manual Operation mode, see “Activating manual tilting,” page 55. The TNC then interpolates the corresponding axes.

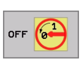
The NC START button has no function. Pressing this button may result in an error message.

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

Switch-off

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

- Select the Manual operating mode



- Select the function for shutting 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

Note

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

To traverse with the machine axis direction buttons:

Select the Manual Operation mode.

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

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

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

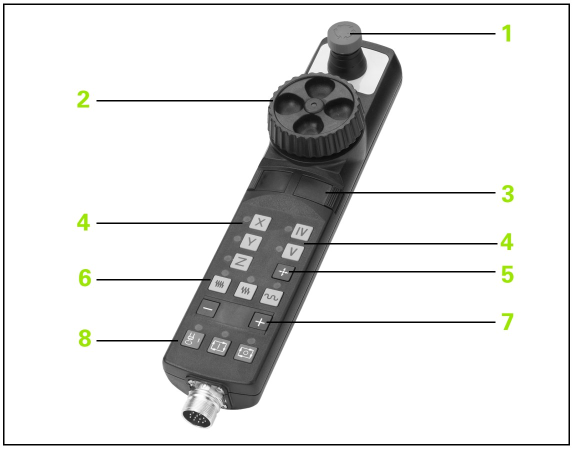
You can move several axes at a time with these two methods. You can change the feed rate at which the axes are traversed with the F soft key (see "Spindle Speed S, Feed Rate F and Miscellaneous Functions M," page 49).

Traversing with the HR 410 electronic handwheel

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

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

The HR 410 handwheel features the following operating elements:



1 EMERGENCY 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 indicator lights show the axis and feed rate you have selected.

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

To move an axis:

Select the Electronic Handwheel operating mode.

Press and hold a permissive button.

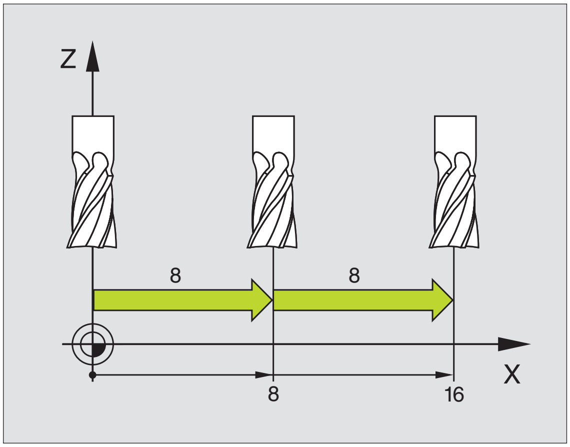
X Select the axis.

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 the 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, i.e. 8 mm.

X

Press the machine axis direction button as often as desired.

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

Function

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

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

Entering values

Spindle speed S, miscellaneous function M

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

SPINDLE SPEED S =

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

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

Feed rate F

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

The following is valid for feed rate F:

- If you enter F=0, then the lowest feed rate from 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 override dial for spindle speed is only functional on machines with infinitely variable spindle drive.

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

Note

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

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

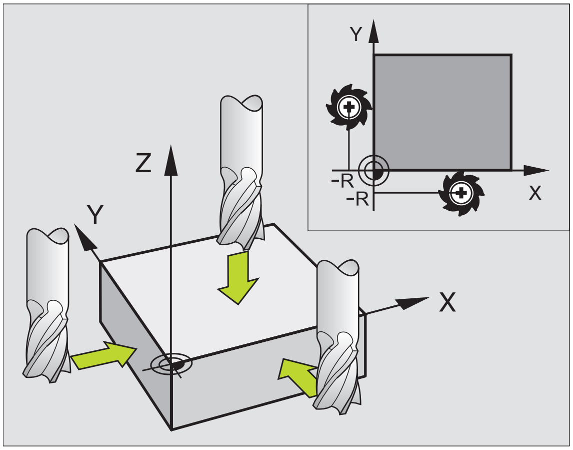
Preparation

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

Datum setting

Fragile workpiece?

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



Select the **Manual Operation** mode.

X Y Z Move the tool slowly until it touches the workpiece surface.

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.

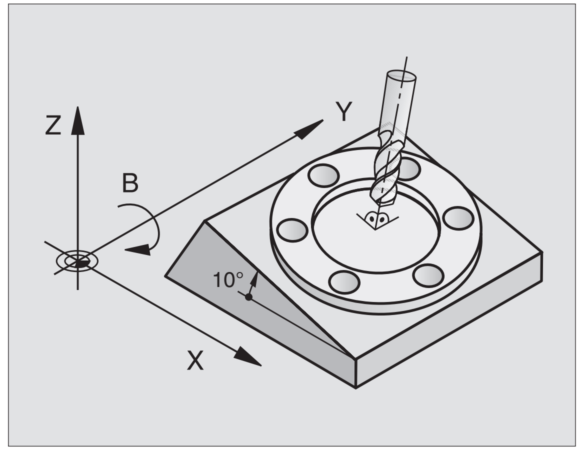
Repeat the process for the remaining axes.

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 Tilting the Working Plane

Application, function

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 rotary axes or as angular components of a tilted plane. Refer to your machine manual.



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, see “Activating manual tilting,” page 55.
- Tilting under program control, Cycle **G80 WORKING PLANE** in the part program (see “WORKING PLANE (Cycle G80)” on page 359).

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:

- **Machine with tilting tables**
  - You must tilt the workpiece into the desired position for machining by positioning the tilting table, for example with a G0 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).

- **Machine with swivel head**
  - You must bring the tool into the desired position for machining by positioning the swivel head, for example with a G0 block.
  - The position of the transformed tool axis changes in relation to the machine-based coordinate system. Thus if you rotate the swivel head of your machine—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 that the actual angle of the tilted axis was entered in the menu field.

Setting the datum in a tilted coordinate system

After you have positioned the rotary axes, set the datum in the same manner 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. Refer to your machine manual.

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.

MP 7500 is effective in the machine parameter list, or, if available, in the descriptive tables for tilted axis geometry. Refer to your machine manual.

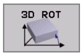
Position display in a tilted system

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.

Activating manual tilting

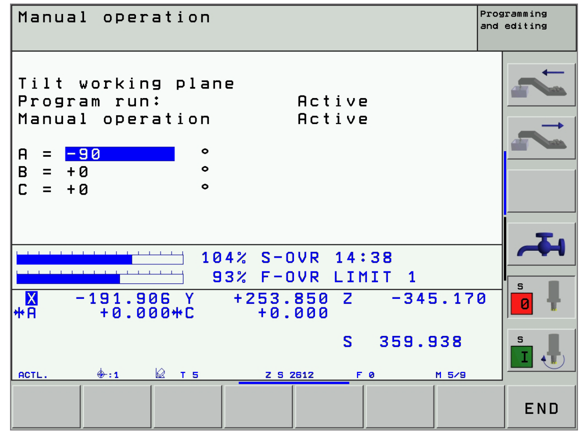


To select manual tilting, press the 3-D ROT soft key. You can now select the desired menu items with the arrow keys.

Enter the tilt angle.

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

To conclude entry, press the END key.



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

If the tilted 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 activate the “Tilt working plane” function for the Program Run operating mode, the tilt angle entered in the menu becomes active in the first block of the part program. If you are using **G80 WORKING PLANE** in the part program, the angular values defined in the cycle (starting at the cycle definition) are effective. Angle values entered in the menu will be overwritten.

3

Positioning with Manual Data Input (MDI)

3.1 Programming and Executing Simple Machining Operations

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

Positioning with Manual Data Input (MDI)

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

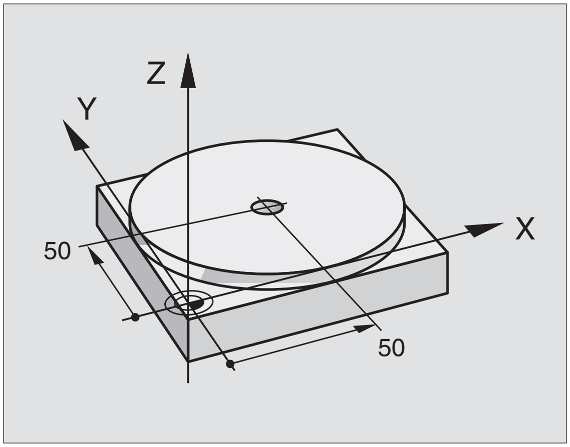
To start program run, press the machine START button.

The following functions are not available:

- Program call with %
- Interactive Programming graphics
- 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 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 **G200** Drilling.

%$MDI G71 *	
N10 G99 T1 L+0 R+5 *	Define tool: zero tool, radius 5
N20 T1 G17 S2000 *	Call tool: tool axis Z
	spindle speed 2000 rpm
N30 G00 G40 G90 Z+200 *	Retract tool (rapid traverse)
N40 X+50 Y+50 M3 *	Move the tool at rapid traverse to a position above
	the hole spindle on
N50 G01 Z+2 F2000 *	Position tool to 2 mm above hole
N60 G200 DRILLING	Define Cycle G200 Drilling
Q200=2 ;SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q201=-20 ;DEPTH	Total hole depth (Algebraic sign=working direction)
Q206=250 ;FEED RATE FOR PLNGNG	Feed rate for pecking
Q202=10 ;PLUNGING DEPTH	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time at top for chip release (in seconds)
Q203=+0 ;SURFACE COORDINATE	Workpiece surface coordinate
Q204=50 ;2ND SET-UP CLEARANCE	Position after the cycle, with respect to Q203
Q211=0.5 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
N70 G79 *	Call Cycle G200 PECKING
N80 G00 G40 Z+200 M2 *	Retract the tool
N9999999 %$MDI G71 *	End of program

Straight-line function **G00** (see "Straight line at rapid traverse G00 Straight line with feed rate G01 F. . ." on page 165), Cycle **G200** Drilling (see "DRILLING (Cycle G200)" on page 225).

Example 2: Correcting workpiece misalignment on machines with rotary tables

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

Write down the rotation angle and cancel the Basic Rotation.

Select operating mode: Positioning with MDI.

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

END Conclude entry.

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

Protecting and erasing programs in $MDI

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

Select the Programming and Editing mode of operation.

PGM MGT

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

Move the highlight to the $MDI file.

COPY ABC → XYZ

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

TARGET FILE =

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

ENT

EXECUTE

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.

If you wish to delete $MDI, then

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

For further information, see "Copying a single file," page 83.

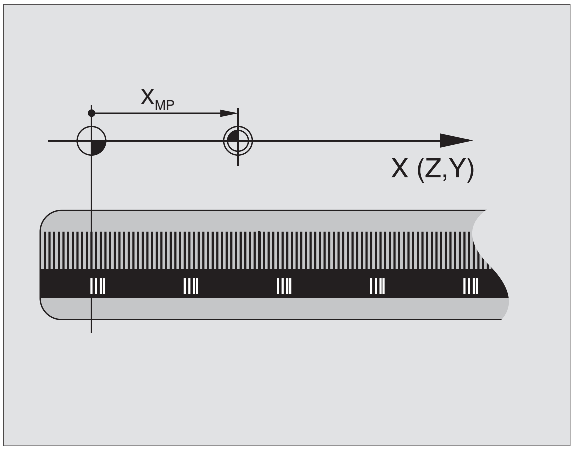
4

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

4.1 Fundamentals

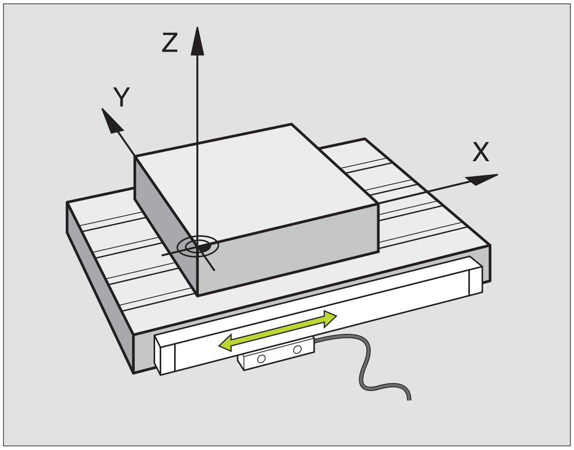
Position encoders and reference marks

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



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

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

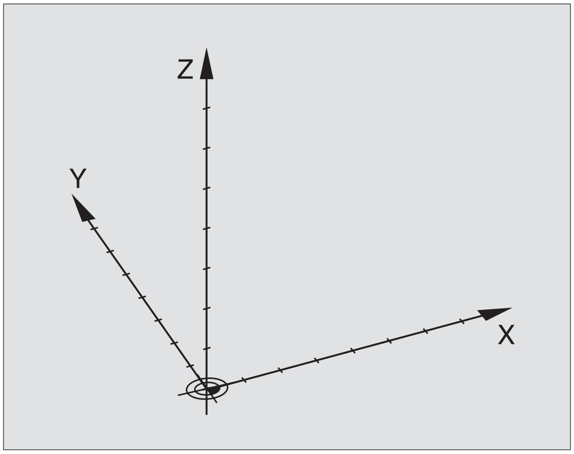


With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.

Reference system

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

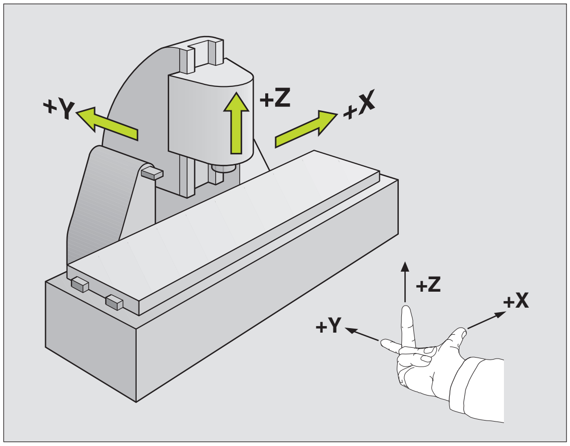
The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is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



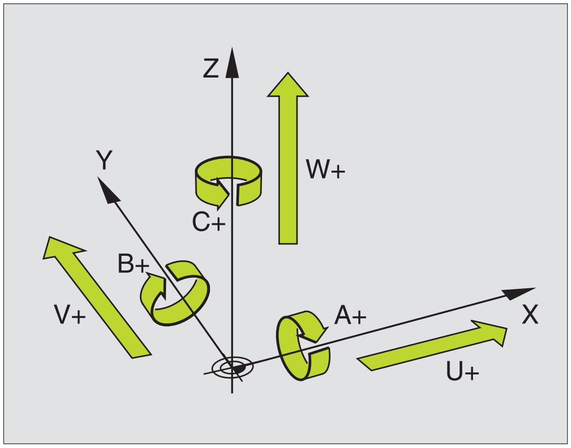
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (reference point) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.

Reference system on milling machines

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



The iTNC 530 can control up to 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.



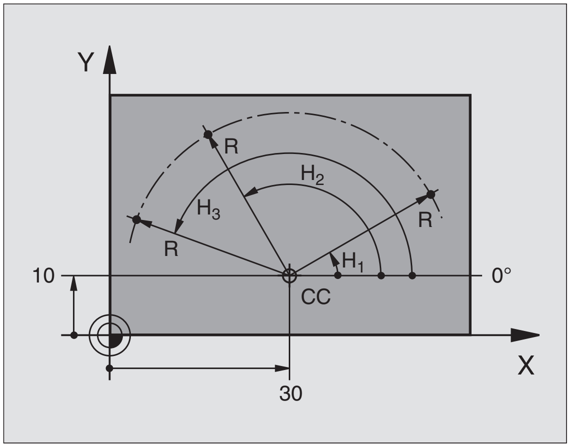
Polar coordinates

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 pole. A position in a plane can be clearly defined by the

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

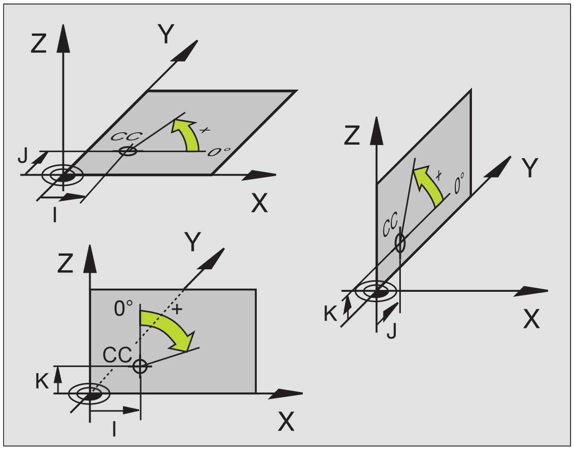
See figure at upper right.



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



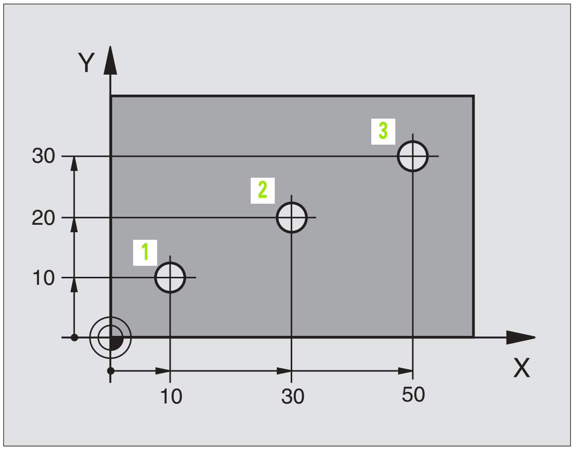
Absolute and incremental workpiece positions

Absolute workpiece positions

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

Example 1: Holes dimensioned in absolute coordinates

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



Incremental workpiece positions

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

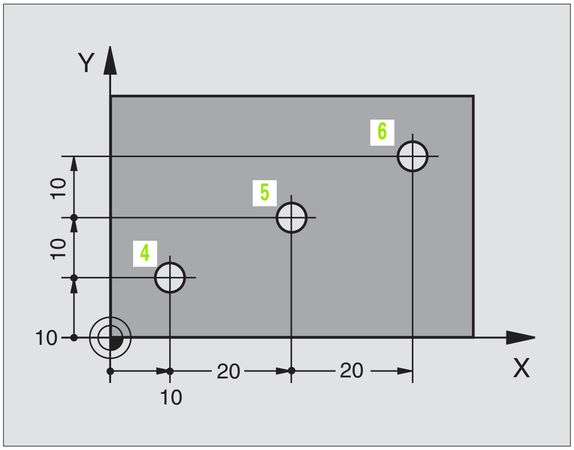
To program a position in incremental coordinates, enter the function G91 before the axis.

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole **4**

X = 10 mm
Y = 10 mm

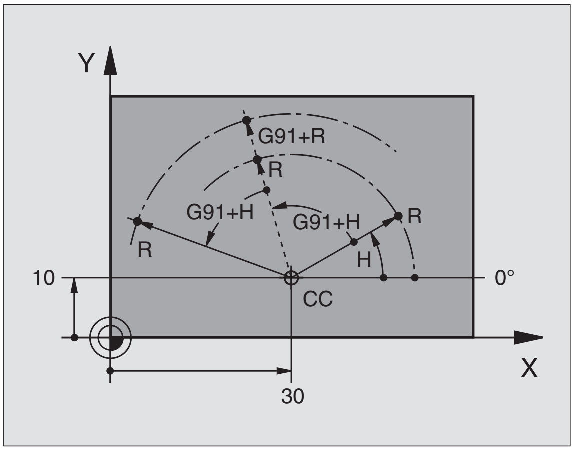
Hole **5**, referenced to **4**	Hole **6**, referenced to **5**
G91 X= 20 mm	G91 X= 20 mm
G91 Y= 10 mm	G91 Y= 10 mm



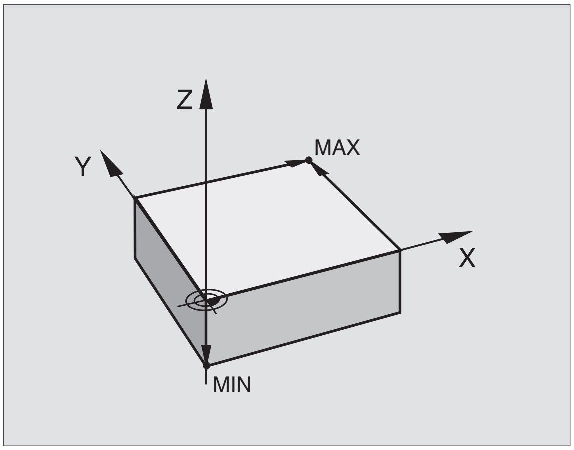
Absolute and incremental polar coordinates

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.



Setting the datum



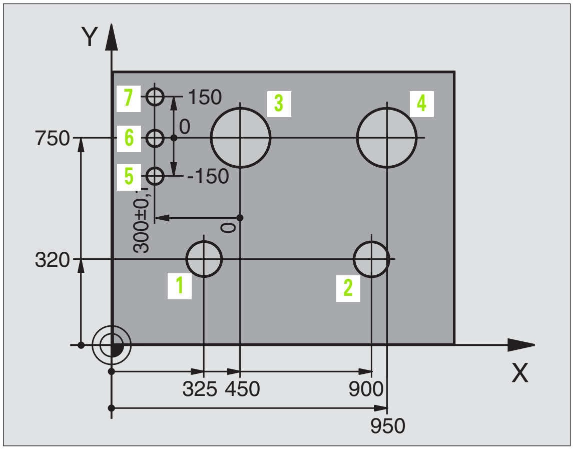
A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. Before setting the datum, you align the workpiece with the machine axes and move the tool in each axis to a known position relative to the workpiece. You then set the TNC display either to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

If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles (see “Coordinate Transformation Cycles” on page 348).

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

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

Example



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

4.2 File Management: Fundamentals

Files

Using the MOD function PGM MGT (see "Configuring PGM MGT" on page 451), select between standard and advanced file management.

If the TNC is connected to a network, then use file management with additional functions.

Files in the TNC	Type
Programs	
In HEIDENHAIN format	.H
In ISO format	.I
Tables for	
Tools	.T
Tool changers	.TCH
Pallets	.P
Datums	.D
Points	.PNT
Cutting data	.CDT
Cutting materials, workpiece materials	.TAB
Texts as	
ASCII files	.A

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

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

You can manage an almost unlimited number of files with the TNC, at least **2000 MB.**

File names

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

PROG20	.I
File name	File type
Maximum Length	See table "Files in the TNC."

Data backup

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

You can do this with the free backup program TNCBACK.EXE from HEIDENHAIN. Your machine tool builder can provide you with a copy of TNCBACK.EXE.

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

Saving the contents of the entire hard disk (> 2 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.

Depending on operating conditions (e.g., vibration load), hard disks generally have a higher failure rate after three to five years of service. HEIDENHAIN therefore recommends having the hard disk inspected after three to five years.

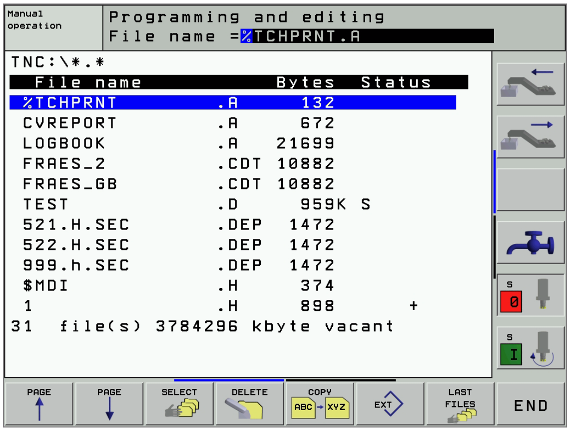
4.3 Standard File Management

Note

> The standard file management is best if you wish to save all files in one directory, or if you are well practiced in the file management of old TNC controls.
>
> To use the standard file management, set the MOD function **PGM MGT** (see "Configuring PGM MGT" on page 451) to **Standard.**

Calling the file manager

Press the PGM MGT key: The TNC displays the file management window (see figure at right)



The window shows you all of the files that are stored in the TNC. Each file is shown with additional information:

Display	Meaning
FILE NAME	Name with up to 16 characters and file type
BYTE	File size in bytes
STATUS	File properties:
E	Program is selected in the Programming and Editing mode of operation.
S	Program is selected in the Test Run mode of operation.
M	Program is selected in a program run mode of operation.
P	File is protected against editing and erasure.

Selecting a file

PGM MGT — Call the file manager.

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

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

PAGE ↓ PAGE ↑ — Moves the highlight up or down **page by page** in the window.

SELECT or ENT — To select the file: Press the SELECT soft key or the ENT key.

Deleting a file

PGM MGT — Call the file manager.

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

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

PAGE ↓ PAGE ↑ — Moves the highlight up or down **page by page** in the window.

DELETE — To delete the file: Press the DELETE soft key.

DELETE ..... FILE?

YES — Confirm with the YES soft key.

NO — Abort with the NO soft key.

Copying a file

PGM MGT — Call the file manager.

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

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

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

To copy the file: Press the COPY soft key.

`TARGET FILE =`

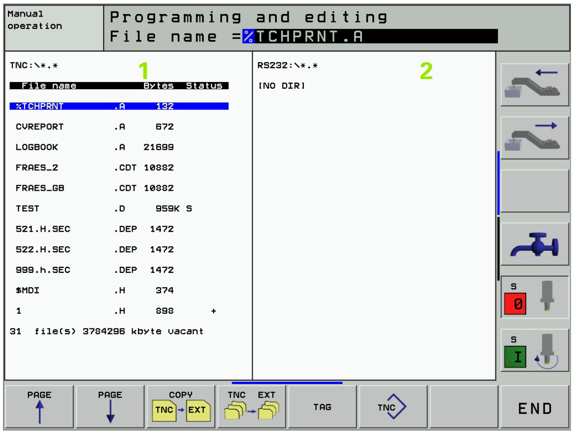
Enter the new name, and confirm your entry with the EXECUTE soft key or the ENT key. A status window appears on the TNC, informing you about the copying progress. As long as the TNC is copying, you can no longer work, or

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.

When the copying process has been started with the EXECUTE soft key, the TNC displays a pop-up window with a progress indicator.

Data transfer to or from an external data medium

Before you can transfer data to an external data medium, you must set up the data interface (see "Setting the Data Interfaces" on page 442).



PGM MGT

Call the file manager.

EXT

Activate data transfer: Press the EXT soft key. In the left half of the screen **(1)** the TNC shows all files saved on its hard disk. In the right half of the screen **(2)** it shows all files saved on the external data medium.

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

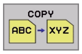
Moves the highlight up and down within a window.

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

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

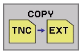
If you wish to copy from the external data medium to the TNC, move the highlight in the right window to the file to be transferred.

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



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

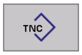
Transfer several files: Press the TAG soft key, or



Transfer all files: Press the TNC => EXT soft key.

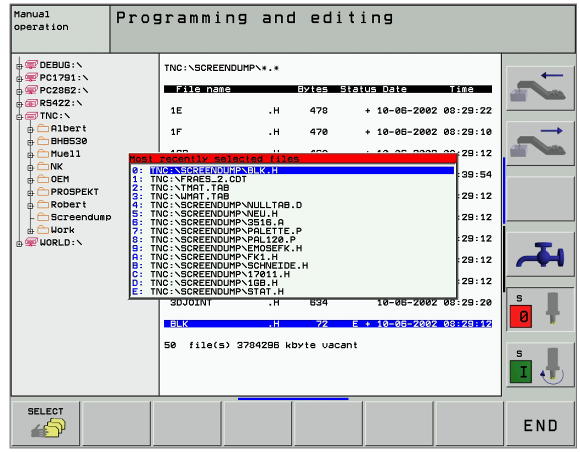
Confirm with the EXECUTE soft key or with the ENT key. A status window appears on the TNC, informing about the copying progress, or

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 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 — Call the file manager.

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

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

↑ ↓ — Move the highlight up or down.

SELECT or ENT — To select the file: Press the SELECT soft key or the ENT key.

Renaming a file

PGM MGT — Call the file manager.

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

↑ ↓ — Moves the highlight up or down **file by file** in the window.

PAGE ↑ PAGE ↓ — Moves the highlight up or down **page by page** in the window.

RENAME ABC = XYZ — Press the RENAME soft key to select the renaming function

TARGET FILE =

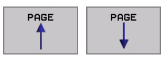
Enter the name of the new file and confirm your entry with the ENT key or EXECUTE soft key.

Protecting a file / Canceling file protection

Call the file manager.

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

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



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

To enable file protection: Press the PROTECT soft key. The file now has status P, or

Press the UNPROTECT soft key to cancel file protection. The P status is canceled.

4.4 Advanced File Management

Note

Use the advanced file manager if you wish to keep your files in individual directories.

To use it, set the MOD function PGM MGT (see “Configuring PGM MGT” on page 451).

See also “File Management: Fundamentals” on page 69.

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 into further directories, which are called subdirectories. With the –/+ key or ENT you can show or hide the 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 16 characters and does not have an extension. If you enter more than 16 characters for the directory name, the TNC will display an error message.

Paths

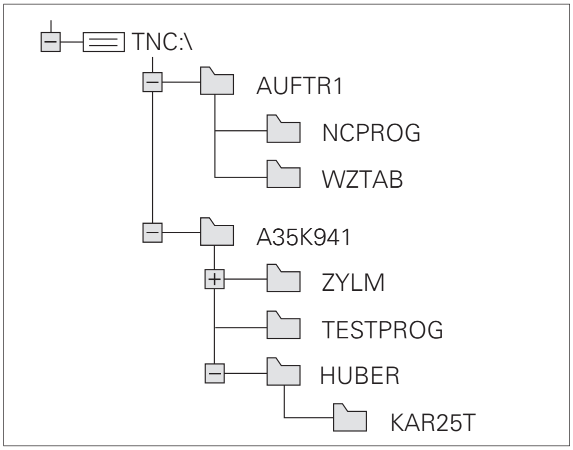
A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash “\”.

Example

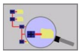
On drive **TNC:** the subdirectory AUFTR1 was created. Then, in the directory **AUFTR1** the directory NCPROG was created and the part program PROG1.H was copied into it. The part program now has the following path:

TNC:\AUFTR1\NCPROG\PROG1.H

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

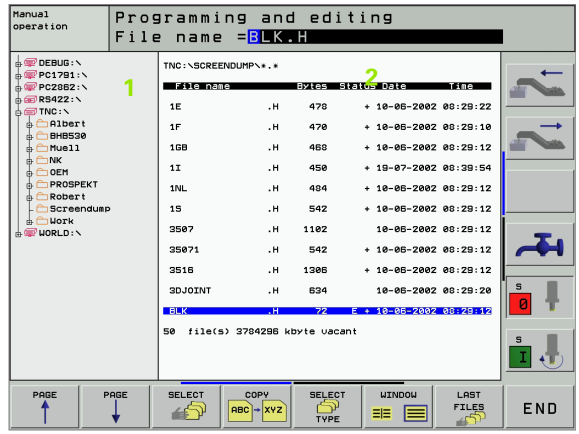


Overview: Functions of the expanded file manager

Function	Soft key
Copy (and convert) individual files	COPY ABC → XYZ
Select target directory	
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
Manage network drives	NET
Copy a directory	COPY DIR
Display all the directories of a particular drive	UPDATE TREE
Delete directory with all its subdirectories	DELETE ALL

Calling the file manager

PGM MGT	Press the PGM MGT soft key: The TNC displays the file management window. (The figure at right shows the basic settings. If the TNC shows a different screen layout, press the WINDOW soft key.)



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

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

Display	Meaning
FILE NAME	Name with up to 16 characters and file type
BYTE	File size in bytes
STATUS	File properties:
E	Program is selected in the Programming and Editing mode of operation.
S	Program is selected in the Test Run mode of operation.
M	Program is selected in a program run mode of operation.
P	File is protected against editing and erasure.
DATE	Date the file was last changed
TIME	Time the file was last changed

Selecting drives, directories and files

PGM MGT — Call the file manager.

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

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

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

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

1st step: Select drive

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

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

2nd step: Select a directory

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

3rd step: select a file

SELECT TYPE

Press the SELECT TYPE soft key.

SHOW .H

Press the soft key for the desired file type, or

SHOW ALL

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

4*.H ENT

Use wild card characters, e.g. to show all files of the file type .H that begin with 4.

Move the highlight to the desired file in the right window

SELECT

or

ENT

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

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

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

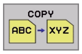
Press the YES soft key to confirm, or

NO

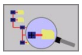
Abort with the NO soft key.

Copying a single file

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

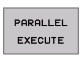


- Press the COPY soft key to select the copy function. The TNC displays a soft-key row with soft keys for different functions.



- Press the “Select target directory” soft key to select the desired directory in a pop-up window. After the target directory has been selected, the corresponding path is indicated in the header. Use the Backspace key to position the cursor directly at the end of the path name and enter the name of the destination file.

- 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 or into the selected destination directory. The original file is retained, or



- 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.

> When the copying process has been started with the EXECUTE soft key, the TNC displays a pop-up window with a progress indicator.

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.

> The **REPLACE FIELDS** soft key does not appear when you want to overwrite the table in the TNC with an external data transfer software, such as TNCremoNT. Copy the externally created file into a different directory, and then copy the desired fields with the TNC file management.

Example

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

- Tool number (column **T**)
- Tool length (column **L**)
- Tool radius (column **R**)

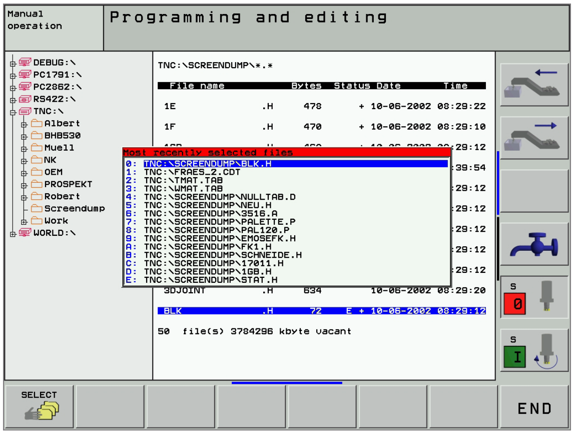
Copy this file to a directory other than the one containing the previous TOOL.T. If you wish to copy this file over the existing table using the TNC file management, the TNC asks if you wish to overwrite the existing TOOL.T tool table:

- 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.
- Or, 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 TNC does not change the data in the other lines and columns.

Copying a directory

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

Choosing one of the last 10 files selected



PGM MGT

Call the file manager.

LAST FILES

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

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

Moves the highlight up and down within a window.

SELECT

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

Deleting a directory

▶ Delete 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

Tagging functions	Soft key
Tag a single file	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

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 tagging 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:

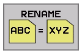
TAG FILE — To mark more files, press the TAG FILE soft key.

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 the DELETE soft key to delete the tagged files.

Renaming a file

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



- 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.

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

- 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.

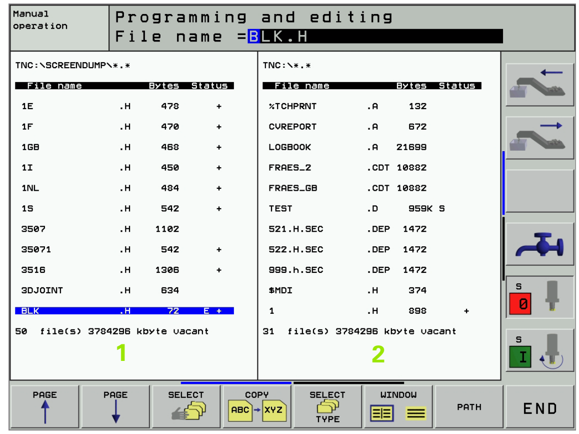
Erasing a directory together with all its subdirectories and files

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

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

- 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 up the data interface (see "Setting the Data Interfaces" on page 442).

PGM MGT — Call the file manager.

WINDOW — Select the screen layout for data transfer: press the WINDOW soft key. In the left half of the screen **(1)** the TNC shows all files saved on its hard disk. In the right half of the screen **(2)** it shows all files saved on the external data medium.

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

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

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

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

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

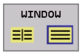
COPY ABC → XYZ — Transfer a single file: Press the COPY soft key, or

TAG — Transfer several files: Press the TAG soft key (in the second soft-key row, see "Tagging files," page 86), or

COPY TNC → EXT — Transfer all files: Press the TNC => EXT soft key.

Confirm with the EXECUTE soft key 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 the left window and then press the WINDOW soft key. The standard file manager window is displayed again.

To select another directory in the split-screen display, press the PATH soft key. Select the desired directory in the pop-up window by 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

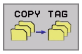
- Move the highlight to the directory into which you wish to copy the files, and display the files in this directory with the ENT key.

In the left window

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

- Display the file tagging functions.

- Move the highlight to the file you want to copy and tag it. You can tag several files in this way, if desired.



- Copy the tagged files into the target directory.

Additional tagging functions: see “Tagging files,” page 86.

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

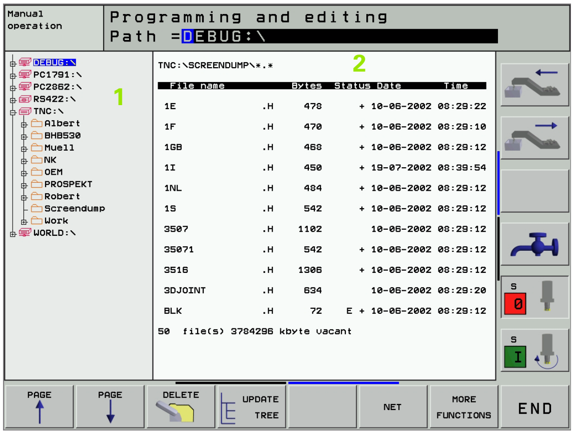
If you copy files into a directory in which other files are stored under the same name, the TNC will ask whether the files in the target directory should be overwritten:

- To overwrite all files, press the YES soft key, or
- To overwrite no files, press the NO soft key, or
- To confirm each file separately before overwriting it, press the CONFIRM soft key.

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

The TNC in a Network

> To connect the Ethernet card to your network, (see "Ethernet Interface" on page 447).
>
> The TNC logs error messages during network operation (see "Ethernet Interface" on page 447).



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

Connecting and disconnecting a network drive

PGM MGT

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

NET

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

Function	Soft key
Establish 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 network connection whenever the TNC is switched on. The TNC shows an **A** in the **Auto** column if the connection is established automatically.	AUTO MOUNT
Do not establish 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 transmission speed is 2 to 5 MB/s, depending on the type of file being transferred and how busy the network is.

4.5 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.

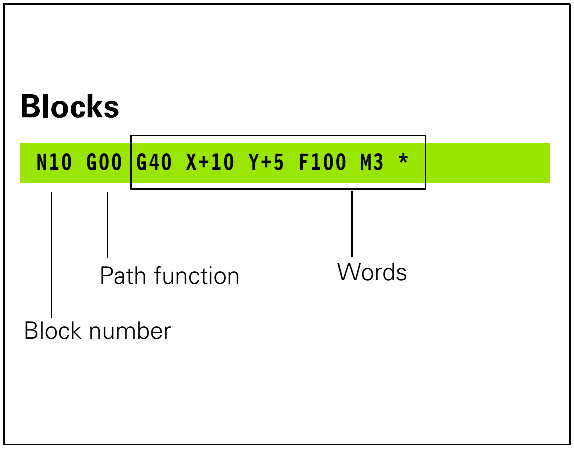
The TNC numbers the blocks of a part program automatically depending on MP7220. MP7220 defines the block number increment.

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

The last block of a program is identified by **N99999999 %,** the program name and the active unit of measure (G70/G71).



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 lie parallel to the X, Y and Z axes and can be up to 100 000 mm long. The blank form is defined by two of its corner points:

- MIN point 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 (with G91).

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

Creating a new part program

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

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.H

ENT

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

MM

To select the unit of measure, press the MM or INCH soft key. The TNC switches the screen layout and initiates the dialog for defining the blank form.

SPINDLE AXIS?

ENT

Set the spindle axis (use the default setting G17 = Z or, if required, select a different spindle axis using the soft key) and confirm with the ENT key.

COORDINATES?

0 ENT

0 ENT

-40 ENT

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

COORDINATES?

G90 G91

Define absolute/incremental input; can be defined separately for each coordinate.

COORDINATES?

100 ENT

100 ENT

0 ENT

Enter in sequence the X, Y and Z coordinates of the MAX point and confirm each of your entries with the ENT key.

Example: Display the blank form in the NC program

`%NEW G71 *`	Program begin, name, unit of measure
`N10 G30 G17 X+0 Y+0 Z-40 *`	Spindle axis, MIN point coordinates
`N20 G31 G90 X+100 Y+100 Z+0 *`	MAX point coordinates
`N9999999 %NEW G71 *`	Program end, name, unit of measure

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

If you do not wish to define a blank form, cancel the dialog at **Spindle axis Z – XY plane** by pressing the DEL key.

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

Programming tool movements

To program a block, select an ISO function key on the alphabetic keyboard. You can also use the gray contouring keys to get the corresponding G code.

You only need to make sure that capitalization is active.

Example of a positioning block

G 1 ENT Start block.

COORDINATES?

X 10 Enter the target coordinate for the X axis.

Y 5 ENT Enter the target coordinate for the Y axis, and go to the next question with ENT.

PATH OF THE CUTTER CENTER

G 40 Select tool movement without radius compensation: Confirm with the ENT key or

G41 G42 To move the tool to the left or to the right of the contour, select function G41 (to the left) or G42 (to the right) by soft key.

FEED RATE ? F=

750 ENT Enter a feed rate of 750 mm/min for this path contour and confirm with the ENT key.

MISCELLANEOUS FUNCTION M?

3 END Enter the desired miscellaneous function (e.g. M3 Spindle ON) and press the END key to terminate and save the block.

M120 Select the miscellaneous function the TNC displays in the soft-key row.

The program-block window displays the following line:

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

Actual position capture

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

- Positioning-block programming.
- Cycle programming.
- Tool definition with function G99.

To transfer the correct position values, proceed as follows:

▶ Place the input box at the position in the block where you want to insert a position value.

▶ Select the actual position capture function: In the soft-key row the TNC displays the axes whose positions can be transferred.

▶ Select the axis: The TNC writes the current position of the selected axis into the active input box.

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

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

Editing a program

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

Function	Soft key/key
Go to previous page	PAGE ↑
Go to next page	PAGE ↓
Go to beginning of program	BEGIN ↑
Go to end of program	END ↓
Change the position of the current block on the screen: Press this soft key to display additional program blocks that are programmed before the current block.	
Change the position of the current block on the screen: Press this soft key to display additional program blocks that are programmed after the current block.	
Move from one block to the next	↓ ↑
Select individual words in a block	← →

Function	Soft key/key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
Delete the selected block	DEL
Insert the block that was last edited or deleted	INSERT LAST NC BLOCK

Inserting blocks at any desired location

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

Editing and inserting words

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

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

Looking for the same words in different blocks

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

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

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

Marking, copying, deleting and inserting program sections

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

To copy a program section, proceed as follows:

- Select the soft-key row 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. For reasons of clarity, the text you have inserted remains highlighted.
- To end the marking function, press the CANCEL SELECTION soft key.

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

Changing the block number increment

If you have deleted, moved or added program sections, you can have the TNC renumber the blocks by pressing the ORDER BLOCK NUMBERS soft key:

ORDER N

- To renumber the blocks, press the ORDER BLOCK NUMBERS soft key. The TNC superimposes a window where you can enter the block number increment.
- Enter the desired block number increment and confirm your entry with the ENT key. The TNC renumbers the complete program.

When the TNC inserts a new NC block, it uses the block number increment that is defined in MP7220.

The TNC search function

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

Searching for texts

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

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

G 40
- Enter the text to be searched for. Please note that the search is case-sensitive.

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

WHOLE WORD OFF ON
- If required, change the search options.

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

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

END
- End the search function.

Search functions	Soft key
Show the superimposed window containing the last search items. Use the arrow keys to select a search item and confirm with the ENT key.	LAST SEARCH ELEMENTS
Show the superimposed window containing possible search items of the current block. Use the arrow keys to select a search item and confirm with the ENT key.	CURRENT BLOCK ELEMENTS
Show the superimposed window containing a selection of the most important NC functions. Use the arrow keys to select a search item and confirm with the ENT key.	NC BLOCKS
Activate the Find/Replace function.	SEARCH + REPLACE

Search options	Soft key
Define the search direction.	UPWARD DOWNWARD / UPWARD DOWNWARD
Define the end of the search: With COMPLETE, the search starts at the current block and is continued until the current block is reached again.	COMPLETE BEGIN/END / COMPLETE BEGIN/END
Start a new search.	NEW SEARCH

Find/Replace any text

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

FIND
- Select the Search function: The TNC superimposes the search window and displays the available search functions in the soft-key row.

SEARCH + REPLACE
- Activate the Replace function: The TNC superimposes a window for entering the text to be inserted.

G 02
- Enter the text to be searched for. Please note that the search is case-sensitive. Then confirm with the ENT key.

G 03
- Enter the text to be inserted. Please note that the entry is case-sensitive.

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

WHOLE WORD OFF ON
- If required, change the search options.

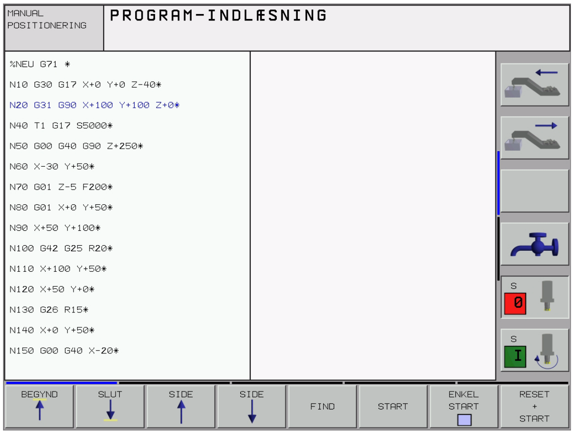
EXECUTE
- Start the search process: The TNC moves to the next occurrence of the text you are searching for.

EXECUTE
- If you wish to replace the text and then move to the next position where the text was found, press the REPLACE soft key. If you do not want to replace the text, but move to the next position where the text was found, press the DO NOT REPLACE soft key.

END
- End the search function.

4.6 Interactive Programming Graphics

To generate/not generate graphics during programming:



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

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

AUTO DRAW OFF ON

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

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

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

Generating a graphic for an existing program

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

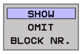
RESET + START

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

Additional functions:

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

Block number display ON/OFF



- Shift the soft-key row.
- To show block numbers: Set the SHOW OMIT BLOCK NR. soft key to SHOW.
- To omit block numbers: Set the SHOW OMIT BLOCK NR. soft key to OMIT.

To erase the graphic:

- Shift the soft-key row.
- Delete graphic: Press CLEAR GRAPHIC soft key.

Magnifying or reducing a detail

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

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

The following functions are available:

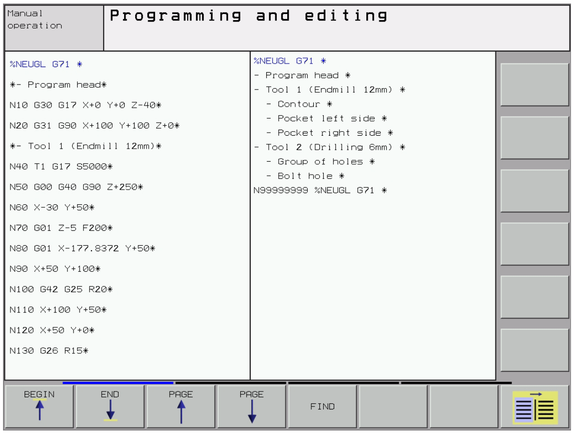
Function	Soft key
Show and move the frame overlay. Press and hold the desired soft key to move the frame overlay.	← → ↓ ↑
Reduce the frame overlay—press and hold the soft key to reduce the detail.	<<
Enlarge the frame overlay—press and hold the soft key to magnify the detail.	>>

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

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

4.7 Structuring Programs

Definition and applications



This TNC function enables you to comment part programs in structuring blocks. Structuring blocks are short texts with up to 244 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex programs in a clear and comprehensible way.

This function is particularly convenient if you want to change the program later. Structuring blocks can be inserted into the part program at any point. They can also be displayed in a separate window, and edited or added to, as desired.

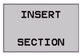
Displaying the program structure window / Changing the active window

▶ To display the program structure window, select the screen display PGM+SECTS.

▶ To change the active window, press the "Change window" soft key.

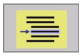
Inserting a structuring block in the (left) program window

▶ Select the block after which the structuring block is to be inserted.



▶ Press the INSERT STRUCTURE soft key or the * key on the ASCII keyboard.

▶ Enter the structuring text with the alphabetic keyboard.



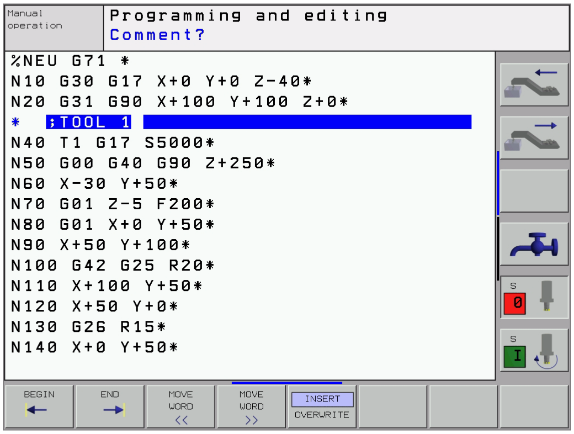
▶ If necessary, change the structure depth with the soft key.

Selecting blocks in the program structure window

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

4.8 Adding Comments

Function



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

Entering comments during programming

- 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.

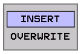
Inserting comments after program entry

- Select the block to which a comment is to be added.
- Select any word in the block with the right arrow key, 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.

Entering 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.

Functions for editing of the comment

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

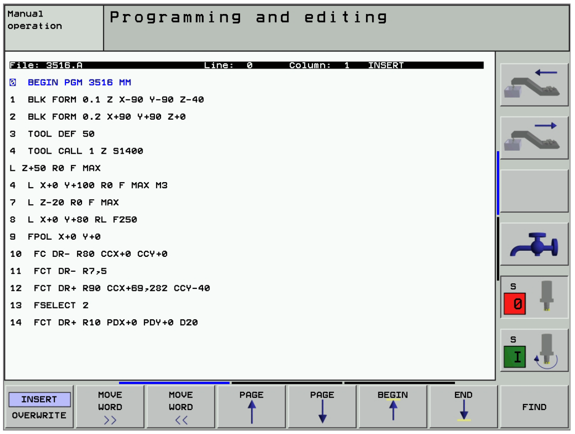
4.9 Creating Text Files

Function

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.

Cursor movements	Soft key
Move one word to the right	MOVE WORD >>
Move one word to the left	MOVE WORD <<
Go to next screen page	PAGE ↓
Go to 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

Editing functions	Key
Insert a blank space	SPACE
Switch between upper and lower case letters	SHIFT SPACE

Editing texts

The first line of the text editor is an information headline displaying 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 by replacing it with 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.

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.

- Move the cursor to the word or line that 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 placed in the buffer memory.
- Move the cursor to the location where you wish insert the text, and press the RESTORE LINE/WORD soft key.

Function	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

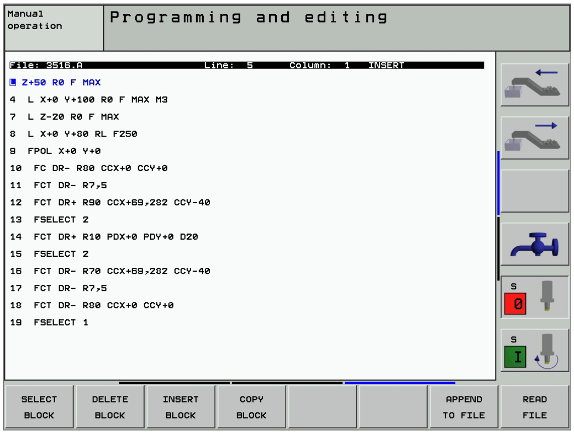
Editing text blocks

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)	INSERT BLOCK

If desired, 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 for the text block to be 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

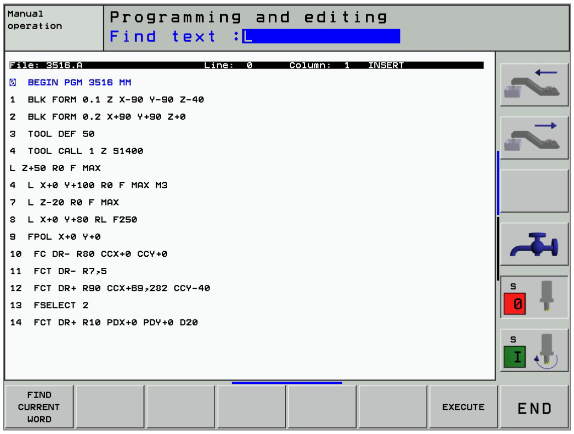
- Move the cursor to the location in the text where you wish to insert another file.

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:



Finding the current text

The search function is used for finding the next occurrence of the word in which the cursor is presently located:

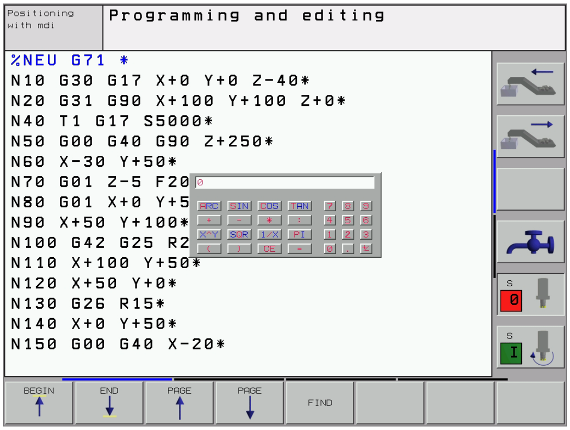
- Move the cursor to the desired word.
- To select the search function, press the FIND soft key.
- Press the FIND CURRENT WORD soft key.
- To leave the search function, press the END soft key.

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 Integrated Pocket Calculator

Operation



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

- Use the CALC key to show and hide the on-line pocket calculator.
- 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 (key)
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	=

To transfer the calculated value into the program,

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

4.11 Immediate Help for NC Error Messages

Displaying error messages

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

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

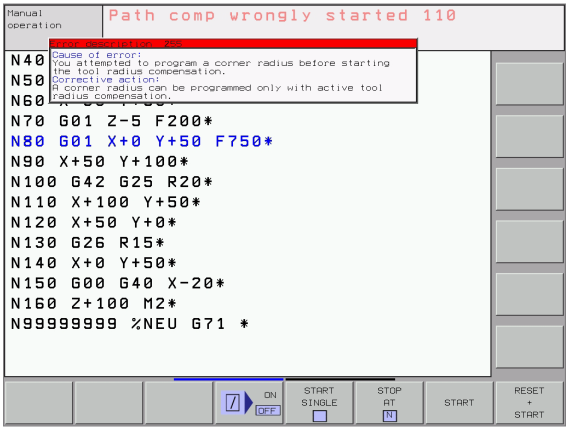
An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. The TNC error messages can be canceled with the CE key, after the cause of the error has been removed.

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

HELP

- ▶ 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 key, thus canceling the error message.
- ▶ Remove the cause of the error as described in the Help window.

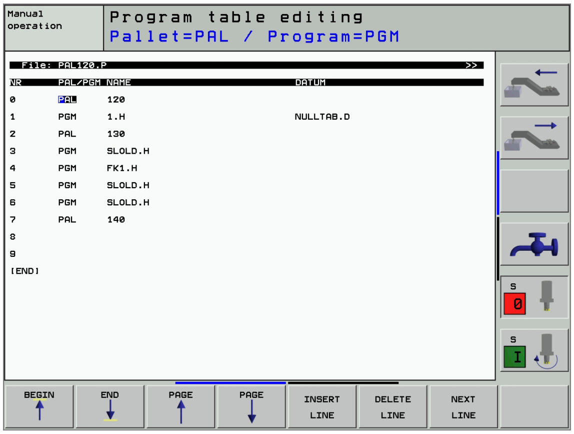


The TNC displays the Help text automatically if the error message is blinking. The TNC needs to be restarted after blinking error messages. To restart the TNC, press and hold the END key for two seconds.

4.12 Pallet Management

Function

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



Pallet tables are used for machining centers with pallet changers: 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 G53 **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 table below):

Position	Meaning
Actual values	Enter the coordinates of the current tool position referenced to the active coordinate system.
Reference values	Enter the coordinates of the current tool position referenced to the machine datum.
ACTL measured values	Enter the coordinates referenced to the active coordinate system of the datum last probed in the Manual operating mode.
REF measured values	Enter the coordinates referenced 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. If you do not define an entry, the datum that was set manually remains active.

Editing 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 as last line in the table	INSERT LINE
Delete the last line in the table	DELETE LINE
Go to beginning of next line	NEXT LINE
Add the entered number of lines at 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

Selecting a pallet table

- Call the file manager in the Programming and Editing or Program Run mode: Press the PGM MGT key.
- To display all type .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.

Leaving the pallet file

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

Executing the pallet file

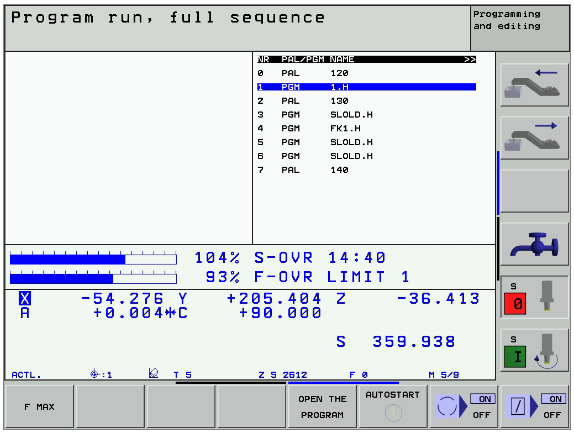
> Programs executed over the pallet file must not contain M30 (M02).
>
> In MP7683, set whether the pallet table is to be executed blockwise or continuously (see "General User Parameters" on page 466).

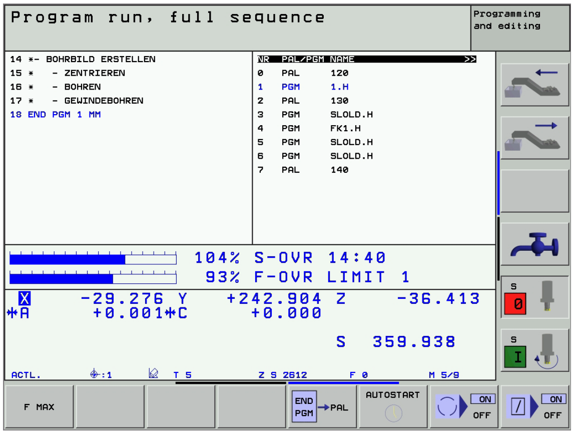
- Select the file manager in the Program Run, Full Sequence or Program Run, Single Block operating modes: Press the PGM MGT key.
- To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- Select the pallet table with the arrow keys and confirm with ENT.
- To execute the pallet table: Press the NC Start button. The TNC executes the pallets as set in MP7683.

Screen layout for executing pallet tables

You can have the TNC display the program contents and pallet file contents on the screen together by selecting the screen layout PGM + PALLET. During execution, the TNC then shows program blocks to the left and the pallet to the right. To check the program contents before execution, proceed as follows:

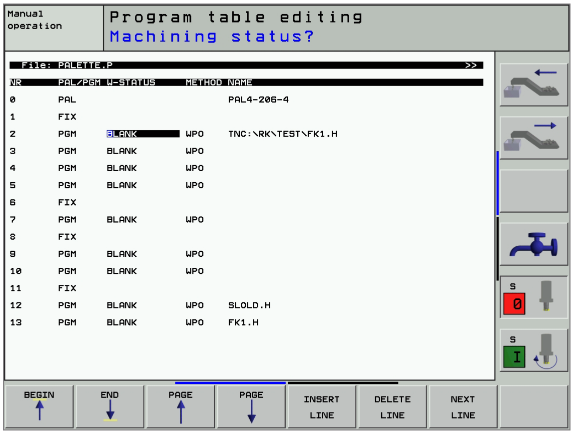
- Select a pallet table.
- With the arrow keys, choose the program you would like to check.
- Press the OPEN PGM soft key: The TNC displays the selected program on the screen. You can now page through the program with the arrow keys.
- To return to the pallet table, press the END PGM soft key.





4.13 Pallet Operation with Tool-Oriented Machining

Function



> Pallet management in combination with tool-oriented machining is a machine-dependent function. The standard functional range will be described below. Refer to your machine manual for more information.

Pallet tables are used for machining centers with pallet changers: 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):
  The entry **PAL** identifies the pallet, **FIX** marks the fixture level and **PGM** is used to enter the workpiece.
- **W-STATE:**
  Current machining status. The machining status is used to determine the current stage of machining. Enter **BLANK** for an unmachined (raw) workpiece. During machining, the TNC changes this entry to **INCOMPLETE,** and after machining has finished, to **ENDED.** The entry **EMPTY** is used to identify a space at which no workpiece is to be clamped or where no machining is to take place.
- **METHOD** (entry obligatory):
  Entry that determines the method of program optimization. Machining is workpiece-oriented if **WPO** is entered. Machining of the piece is tool-oriented if **TO** is entered. In order to include subsequent workpieces in the tool-oriented machining, you must enter **CTO** (continued tool oriented). Tool-oriented machining is also possible with pallet fixtures, but not for multiple pallets.
- **NAME** (entry obligatory):
  Pallet or program name. The machine tool builder determines the pallet name (see Machine Manual). Programs must be stored in the same directory as the pallet table. Otherwise you must enter the full path and 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 G53 **DATUM SHIFT.**
- **X, Y, Z** (entry optional, other axes also possible):
  For pallets and fixtures, the programmed coordinates are referenced to the machine datum. For NC programs, the programmed coordinates are referenced to the pallet or fixture 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 table below):

Position	Meaning
Actual values	Enter the coordinates of the current tool position referenced to the active coordinate system.
Reference values	Enter the coordinates of the current tool position referenced to the machine datum.
ACTL measured values	Enter the coordinates referenced to the active coordinate system of the datum last probed in the Manual operating mode.
REF measured values	Enter the coordinates referenced 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. If you do not define an entry, the datum that was set manually remains active.

- **SP-X, SP-Y, SP-Z** (entry optional, other axes also possible):
  Safety positions can be entered for the axes. These positions can be read with SYSREAD FN18 ID510 NR 6 from NC macros. SYSREAD FN18 ID510 NR 5 can be used to determine if a value was programmed in the column. The positions entered are only approached if these values are read and correspondingly programmed in the NC macros.

- **CTID** (entered by the TNC):
  The context ID number is assigned by the TNC and contains instructions about the machining progress. Machining cannot be resumed if the entry is deleted or changed.

Editing function in table mode	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 as last line in the table	INSERT LINE
Delete the last line in the table	DELETE LINE
Go to beginning of next line	NEXT LINE
Add the entered number of lines at 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

Editing function in entry-form mode	Soft key
Select previous pallet	PALLET ↑
Select next pallet	PALLET ↓
Select previous fixture	FIXTURE ↑
Select next fixture	FIXTURE ↓
Select previous workpiece	WORKPIECE ↑
Select next workpiece	WORKPIECE ↓

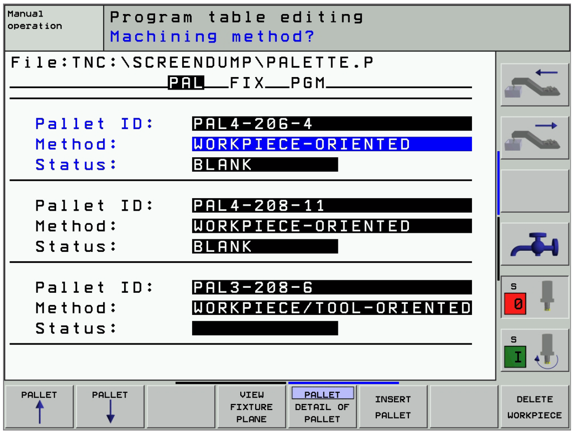
Editing function in entry-form mode	Soft key
Switch to pallet plane	VIEW PALLET PLANE
Switch to fixture plane	VIEW FIXTURE PLANE
Switch to workpiece plane	VIEW WORKPIECE PLANE
Select standard pallet view	PALLET DETAIL OF PALLET
Select detailed pallet view	PALLET DETAIL OF PALLET
Select standard fixture view	FIXTURE DETAIL OF FIXTURE
Select detailed fixture view	FIXTURE DETAIL OF FIXTURE
Select standard workpiece view	WORKPIECE DETAIL OF WORKPIECE
Select detailed workpiece view	WORKPIECE DETAIL OF WORKPIECE
Insert pallet	INSERT PALLET
Insert fixture	INSERT FIXTURE
Insert workpiece	INSERT WORKPIECE
Delete pallet	DELETE PALLET
Delete fixture	DELETE FIXTURE
Delete workpiece	DELETE WORKPIECE
Copy all fields to clipboard	COPY ALL FIELDS
Copy highlighted field to clipboard	COPY SELECTED FIELD
Insert the copied field	PASTE FIELDS
Delete clipboard contents	ERASE INTERMED. MEMORY

Editing function in entry-form mode	Soft key
Tool-optimized machining	TOOL ORIENTAT.
Workpiece-optimized machining	WORKPIECE ORIENTAT.
Connect or separate the types of machining	CONNECTED DIS- CONNECTED
Mark plane as being empty	EMPTY POSITION
Mark plane as being unmachined	BLANK

Selecting a pallet file

- Call the file manager in the Programming and Editing or Program Run mode: Press the PGM MGT key.
- To display all type .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.

Setting up the pallet file with the entry form



Pallet operation with tool- or workpiece-oriented machining is divided into three levels:

- Pallet level **PAL**
- Fixture level **FIX**
- Workpiece level **PGM**

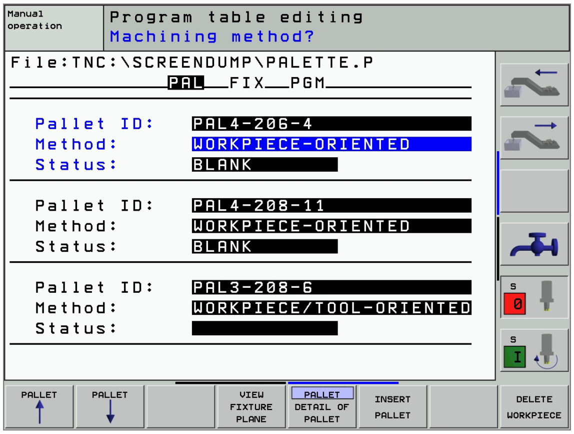
You can switch to a detail view in each level. Set the machining method and the statuses for the pallet, fixture and workpiece in the standard view. If you are editing an existing pallet file, the updated entries are displayed. Use the detail view for setting up the pallet file.

> Set up the pallet file according to the machine configuration. If you only have one fixture with multiple workpieces, then defining one fixture **FIX** with the workpieces **PGM** is sufficient. However, if one pallet contains several fixtures, or if a fixture is machined from more than one side, you must define the pallet **PAL** with the corresponding fixture levels **FIX.**
>
> Use the screen layout button to switch between table view and form view.
>
> Graphic support for form entry is not yet available.

The various levels of the entry form can be reached with the appropriate soft keys. The current level is highlighted in the status line of the entry form. When you switch to table view with the screen layout button, the cursor is placed in the same level as it was in the form view.

Setting up the pallet plane

- **`Pallet Id:`** The pallet name is displayed
- **`Method`**: You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. The selected method is assumed for the workpiece level and overwrites any existing entries. In tabular view, WORKPIECE ORIENTED appears as **`WPO,`** and TOOL ORIENTED appears as **`TO.`**

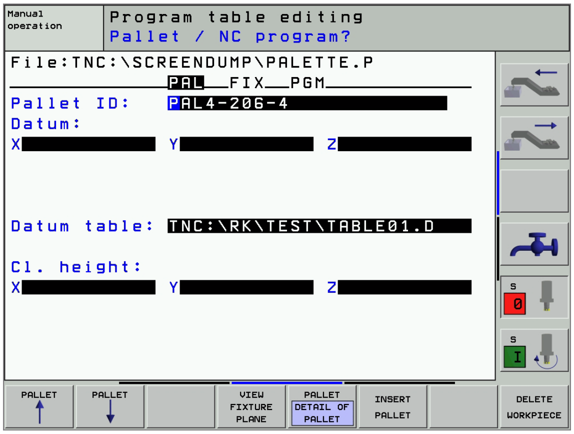


> The TO-/WP-ORIENTED entry cannot be made via soft key. It only appears when different machining methods were chosen for the workpieces in the workpiece or machining level.
>
> If the machining method was determined in the fixture level, the entries are transferred to the workpiece level, where they overwrite any existing entries.

- **`Status:`** The soft key **`BLANK`** identifies the pallet and the corresponding fixtures and workpieces as not yet having been machined, and enters **`BLANK`** in the Status field. Use the soft key **`EMPTY POSITION`** if you want to skip the pallet during machining. **`EMPTY`** appears in the Status field.

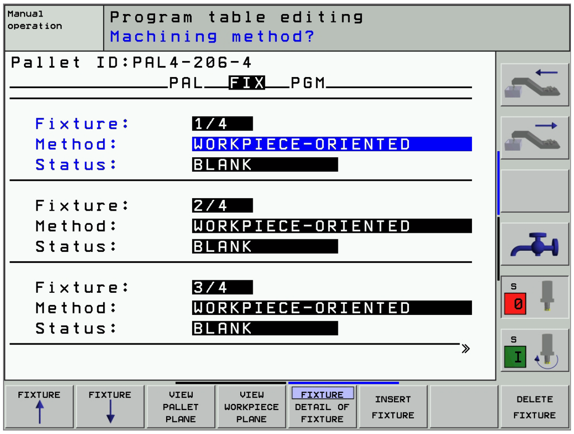
Setting up details in the pallet level

- **`Pallet ID:`** Enter the pallet name.
- **`Datum:`** Enter the pallet datum.
- **`Datum table:`** Enter the name and path of the datum table of the workpiece. The data is transferred to the fixture and workpiece levels.
- **`Safe height:`** (optional): Safe position for the individual axes referenced to the pallet. The positions entered are only approached if these values were read and correspondingly programmed in the NC macros.



Setting up the fixture level

- **`Fixture:`** The number of the fixture is displayed. The number of fixtures within this level is shown after the slash.
- **`Method`**: You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. The selected method is assumed for the workpiece level and overwrites any existing entries. In tabular view, WORKPIECE ORIENTED appears as **`WPO`**, and TOOL ORIENTED appears as **`TO`**.
  Use the **`CONNECT/SEPARATE`** soft key to mark fixtures that are to be included for calculating the machining process for tool-oriented machining. Connected fixtures are marked with a dashed line, whereas separated fixtures are connected with a solid line. Connected workpieces are marked in tabular view with the entry **`CTO`** in the METHOD column.

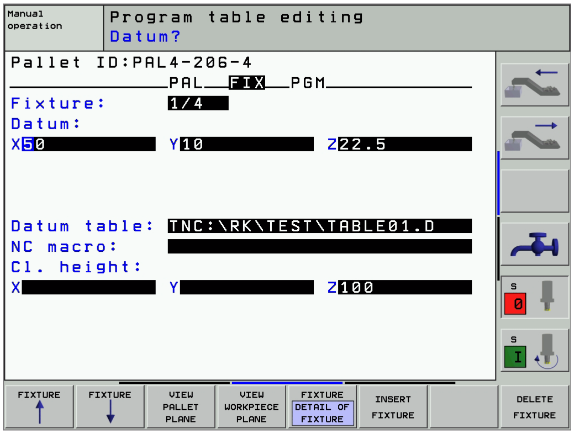


> The TO-/WP-ORIENTED entry cannot be made via soft key. It only appears when different machining methods were chosen for the workpieces in the workpiece level.
>
> If the machining method was determined in the fixture level, the entries are transferred to the workpiece level, where they overwrite any existing entries.

- **`Status:`** The soft key **`BLANK`** identifies the fixture and the corresponding workpieces as not yet having been machined, and enters BLANK in the Status field. Use the soft key **`EMPTY POSITION`** if you want to skip the fixture during machining. **`EMPTY`** appears in the Status field.

Setting up details in the fixture level

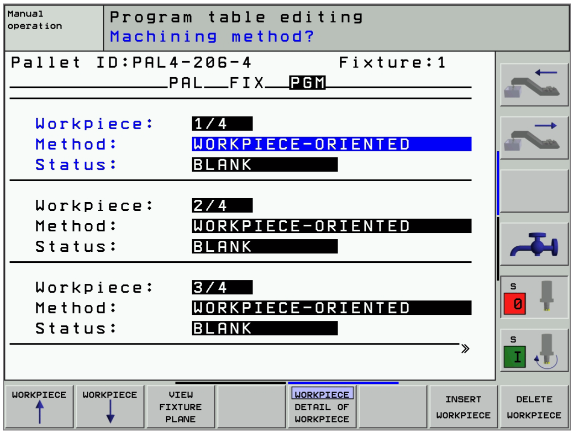
- **`Fixture:`** The number of the fixture is displayed. The number of fixtures within this level is shown after the slash.
- **`Datum:`** Enter the fixture datum.
- **`Datum table:`** Enter the name and path of the datum table valid for machining the workpiece. The data is transferred to the workpiece level.
- **`NC macro:`** In tool-oriented machining, the macro TCTOOLMODE is carried out instead of the normal tool-change macro.
- **`Safe height:`** (optional): Safe position for the individual axes referenced to the fixture.



> Safety positions can be entered for the axes. These positions can be read with SYSREAD FN18 ID510 NR 6 from NC macros. SYSREAD FN18 ID510 NR 5 can be used to determine if a value was programmed in the column. The positions entered are only approached if these values are read and correspondingly programmed in the NC macros.

Setting up the workpiece level

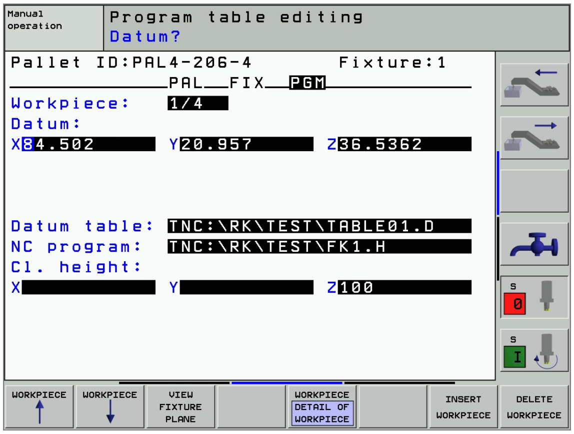
- **`Workpiece:`** The number of the workpiece is displayed. The number of workpieces within this fixture level is shown after the slash.
- **`Method:`** You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. In tabular view, WORKPIECE ORIENTED appears as **`WPO`**, and TOOL ORIENTED appears as **`TO`**.
  Use the **`CONNECT/SEPARATE`** soft key to mark workpieces that are to be included for calculating the machining process for tool-oriented machining. Connected workpieces are marked with a dashed line, whereas separated workpieces are connected with a solid line. Connected workpieces are marked in tabular view with the entry **`CTO`** in the METHOD column.
- **`Status:`** The soft key **`BLANK`** identifies the workpiece as not yet having been machined, and enters BLANK in the Status field. Use the soft key **`EMPTY POSITION`** if you want to skip the workpiece during machining. EMPTY appears in the Status field.



> Enter the method and status in the pallet or fixture level. Then the entry will be assumed for all corresponding workpieces.
>
> For several workpiece variants within one level, the workpieces of one variant should be entered together. This way, the workpieces of each variant can be marked with the CONNECT/SEPARATE soft key, and can be machined in groups.

Setting up details in the workpiece level

- **`Workpiece:`** The number of the workpiece is displayed. The number of workpieces within this fixture or pallet level is shown after the slash.
- **`Datum:`** Enter the workpiece datum.
- **`Datum table:`** Enter the name and path of the datum table valid for machining the workpiece. If you use the same datum table for all workpieces, enter the name and path in the pallet or fixture levels. The data is automatically transferred to the workpiece level.
- **`NC program:`** Enter the path of the NC program that is necessary for machining the workpiece.
- **`Safe height:`** (optional): Safe position for the individual axes referenced to the workpiece. The positions entered are only approached if these values were read and correspondingly programmed in the NC macros.



Sequence of tool-oriented machining

> The TNC only carries out tool-oriented machining if the TOOL ORIENTED method was selected, and TO or CTO is entered in the table.

- The entry TO or CTO in the Method field tells the TNC that the oriented machining is valid beyond these lines.
- The pallet management starts the NC program given in the line with the entry TO.
- The first workpiece is machined until the next tool call is pending. Departure from the workpiece is coordinated by a special tool-change macro.
- The entry in the column W-STATE is changed from BLANK to INCOMPLETE, and the TNC enters a hexadecimal value in the field CTID.

> The value entered in the field CTID is a unique identifier of the machining progress for the TNC. If this value is deleted or changed, machining cannot be continued, nor is mid-program startup or resumption of machining possible.

- All lines in the pallet file that contain the entry CTO in the Method field are machined in the same manner as the first workpiece. Workpieces in several fixtures can be machined.
- The TNC uses the next tool for the following machining steps again from the line with the entry TO if one of the following situations applies:
  - If the entry PAL is in the PAL/PGM field in the next line.
  - If the entry TO or WPO is in the Method field in the next line.
  - If in the lines already machined there are entries under Method which do not have the status EMPTY or ENDED.
- The NC program is continued at the stored location based on the value entered in the CTID field. Usually the tool is changed for the first piece, but the TNC suppresses the tool change for the following workpieces.
- The entry in the CTID field is updated after every machining step. If an END PGM or M02 is executed in an NC program, then an existing entry is deleted and ENDED is entered in the Machining Status field.

- If the entries TO or CTO for all workpieces within a group contain the status ENDED, the next lines in the pallet file are run.

In mid-program startup, only one tool-oriented machining operation is possible. Subsequent pieces are machined according to the method entered.

The value entered in the CTID field is stored for a maximum of one week. Within this time the machining process can be continued at the stored location. After this time the value is deleted, in order to prevent large amounts of unnecessary data on the hard disk.

The operating mode can be changed after executing a group of entries with TO or CTO.

The following functions are not permitted:

- Switching the traverse range
- PLC datum shift
- M118

Leaving the pallet file

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

Executing the pallet file

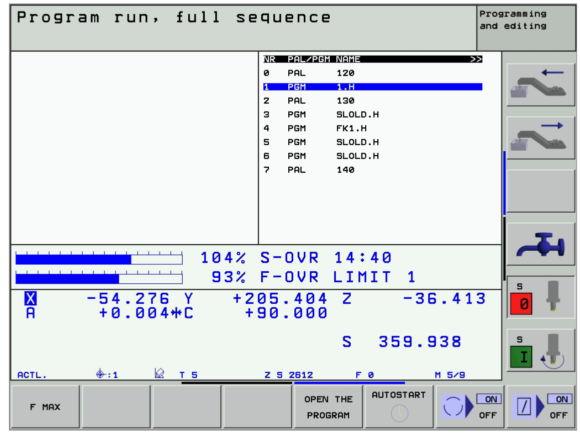
In MP7683, set whether the pallet table is to be executed blockwise or continuously (see “General User Parameters” on page 466).

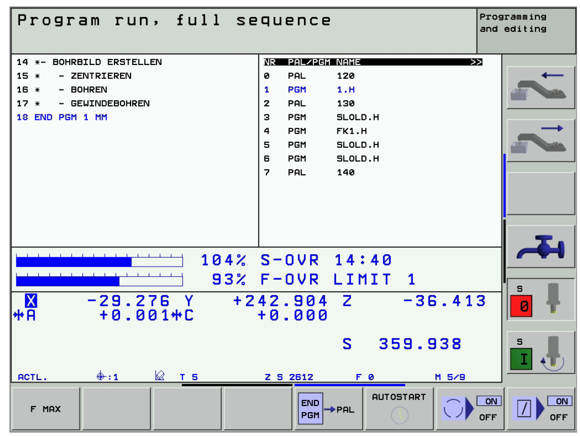
- Select the file manager in the Program Run, Full Sequence or Program Run, Single Block operating modes: Press the PGM MGT key.
- To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- Select the pallet table with the arrow keys and confirm with ENT.
- To execute the pallet table: Press the NC Start button. The TNC executes the pallets as set in MP7683.

Screen layout for executing pallet tables

You can have the TNC display the program contents and pallet file contents on the screen together by selecting the screen layout PGM + PALLET. During execution, the TNC then shows program blocks to the left and the pallet to the right. To check the program contents before execution, proceed as follows:

- Select a pallet table.
- With the arrow keys, choose the program you would like to check.
- Press the OPEN PGM soft key: The TNC displays the selected program on the screen. You can now page through the program with the arrow keys.
- To return to the pallet table, press the END PGM soft key.



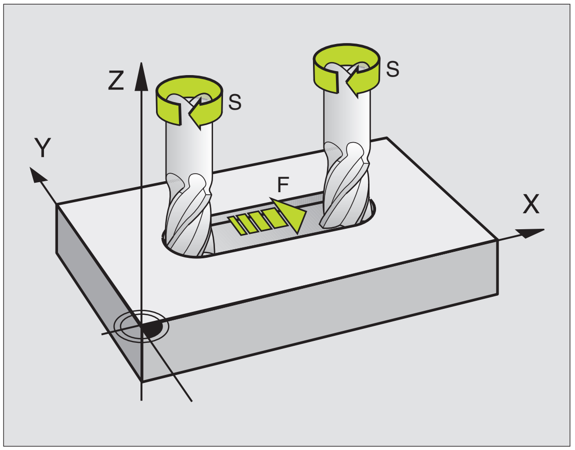


5

Programming: Tools

5.1 Entering Tool-Related Data

Feed rate F



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

Input

You can enter the feed rate in 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

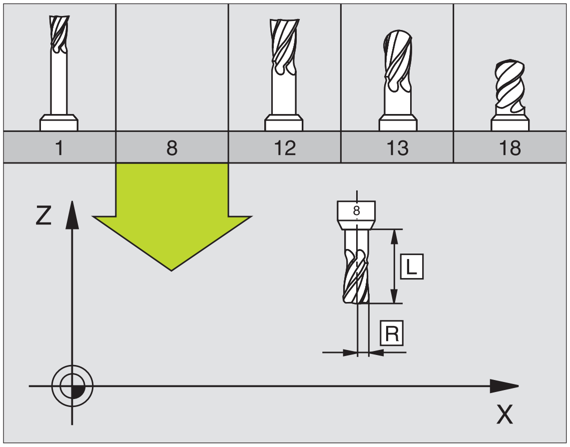
- Press the S key on the alphabetic keyboard.
- Enter the new spindle speed.

Changing during program run

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

5.2 Tool Data

Requirements for tool compensation



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

Tool data can be entered either directly in the part program with **G99** or separately in tool tables. In a tool table, you can also enter additional data for the specific tool. The TNC will consider all the data entered for the tool when executing the part program.

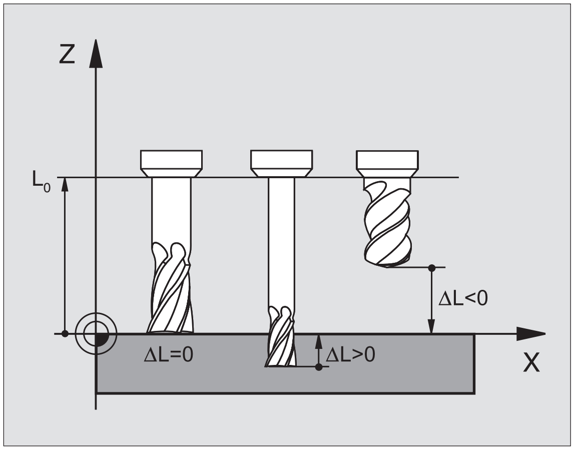
Tool numbers and tool names

Each tool is identified by a number between 0 and 254. If you are working with tool tables, you can use higher numbers and you can also enter a tool name for each tool.

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:

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

For the algebraic sign:

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

To determine the length:

- Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z=0).
- Set the datum in the tool axis to 0 (datum setting).
- Insert the desired tool.
- Move the tool to the same reference position as the zero tool.
- The TNC displays the difference between the current tool and the zero tool.
- Enter the value in the G99 block or in the tool table by pressing the actual-position-capture key.

Determining the length L with a tool presetter

Enter the determined value directly in the **G99** tool definition block or in the tool table without further calculations.

Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii

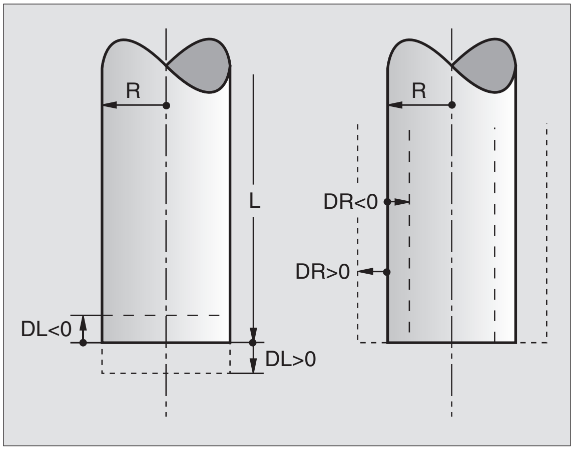
Delta values are offsets in the length and radius of a tool.

A positive delta value describes a tool oversize (DL, DR>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 (DL, 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.



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

Entering tool data into the program

The number, length and radius of a specific tool is defined in the **G99** block of the part program.

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

TOOL DEF

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

▶ **Tool length** : Compensation value for the tool length

▶ **Tool radius** : Compensation value for the tool radius

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

Resulting NC block:

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

Entering tool data in tables

You can define and store up to 32 767 tools and their tool data in a tool table. In Machine Parameter 7260, you can define how many tools are to be stored by the TNC when a new table is set up. Also see the Editing Functions later in this Chapter. In order to be able to assign various compensation data to a tool (indexing the tool number), MP7262 must not be equal to 0.

You must use tool tables if

- you wish to use indexed tools such as stepped drills with more than one length compensation value,
- your machine tool has an automatic tool changer,
- you want to measure tools automatically with the TT 130 touch probe (see the Touch Probe Cycles User's Manual, Chapter 4),
- you want to rough-mill the contour with Cycle **G122** (see "ROUGH-OUT (Cycle G122)" on page 313),
- you want to work with automatic cutting data calculations.

Tool table: Standard tool data

Abbr.	Input	Dialog
T	Number by which the tool is called in the program (e.g. 5, indexed: 5.2)	–
NAME	Name by which the tool is called in the program	**Tool name?**
L	Value for tool length compensation L	**Tool length?**
R	Compensation value for the tool radius R	**Tool radius R?**
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with spherical or toroid cutters)	**Tool radius R2?**
DL	Delta value for tool radius R2	**Tool length oversize?**
DR	Delta value for tool radius R	**Tool radius oversize R?**
DR2	Delta value for tool radius R2	**Tool radius oversize R2?**
LCUTS	Tooth length of the tool for Cycle 22	**Tooth length in the tool axis?**
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	**Maximum plunge angle?**
TL	Set tool lock (**TL: T**ool **L**ocked)	**Tool locked?** **Yes = ENT / No = NO ENT**
RT	Number of a replacement tool (**RT**), if available (also see TIME2)	**Replacement tool?**
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information on TIME1.	**Maximum tool age?**

Abbr.	Input	Dialog
TIME2	Maximum tool life in minutes during a tool call: If the current tool age exceeds this value, the TNC changes the tool during the next tool call (see also CUR.TIME).	**Maximum tool age for TOOL CALL?**
CUR.TIME	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?**
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?**
PLC VAL	Value of this tool that is to be sent to the PLC	**PLC value?**
PTYP	Tool type for evaluation in the pocket table	**Tool type for pocket table?**

Tool table: Tool data required for automatic tool measurement

For a description of the cycles governing automatic tool measurement, see the Touch Probe Cycles Manual, Chapter 4.

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 (M3 = –)?**
TT:R-OFFS	For tool length measurement: tool offset between stylus center and tool center. Preset value: Tool radius R (NO ENT means **R**).	**Tool offset: radius?**
TT:L-OFFS	Tool radius measurement: tool offset in addition to MP6530 (see "General User Parameters" on page 466) between upper surface of stylus and lower surface of tool. Default: 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?**

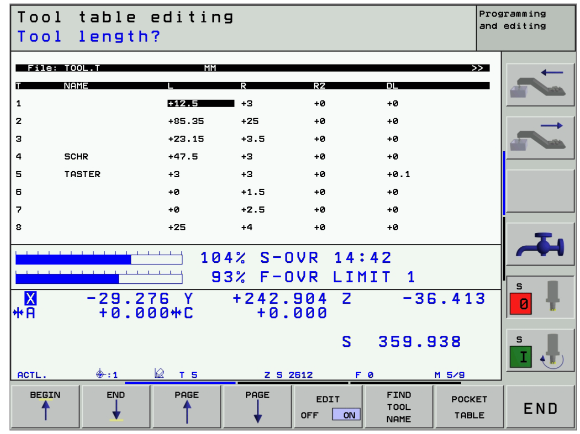
Tool table: Tool data for automatic speed/feed rate calculation.

Abbr.	Input	Dialog
`TYPE`	Tool type (**`MILL`** for milling, **`DRILL`** for drilling or boring, TAP for tapping): Press the SELECT TYPE soft key (3rd soft-key row): The TNC superimposes a window where you can select the type of tool you want.	`Tool type?`
`TMAT`	Tool material: Press the SELECT MATERIAL soft key (3rd soft-key row): The TNC superimposes a window where you can select the type of material you want.	`Tool material?`
`CDT`	Cutting data table: Press the SELECT CDT soft key (3rd soft-key row): The TNC superimposes a window where you can select a cutting data table.	`Name of cutting data table?`

Tool table: Tool data for 3-D touch trigger probe (only when bit 1 is set in MP7411 = 1, also see the Touch Probe Cycles Manual)

Abbr.	Input	Dialog
`CAL-OF1`	During calibration, the TNC stores in this column the center misalignment in the reference axis of the 3-D probe, if a tool number is indicated in the calibration menu.	`Center misalignmt. in ref. axis?`
`CAL-OF2`	During calibration, the TNC stores in this column the center misalignment in the minor axis of the 3-D probe, if a tool number is indicated in the calibration menu.	`Center misalignment minor axis?`
`CAL-ANG`	During calibration, the TNC stores in this column the spindle angle at which the 3-D probe was calibrated, if a tool number is indicated in the calibration menu.	`Spindle angle for calibration?`

Editing tool tables



The tool table that is active during execution of the part program is designated as TOOL.T. TOOL.T must be saved in the directory TNC:\ and can only be edited in one 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

- Call the file manager.
- To select the file type, press the SELECT TYPE soft key.
- To show type .T files, press the SHOW .T soft key.
- Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key.

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

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

Editing functions for tool tables	Soft key
Select beginning of table	BEGIN ↑
Select end of table	END ↓
Select previous page in table	PAGE ↑
Select next page in table	PAGE ↓
Look for the tool name in the table	FIND TOOL NAME
Show tool information in columns or show all information on one tool on one screen page	LIST FORM

Editing functions for tool tables	Soft key
Move to beginning of line	BEGIN LINE ←
Move to end of line	END LINE →
Copy highlighted field	COPY FIELD
Insert copied field	PASTE FIELD
Add the entered number of lines (tools) at the end of the table.	APPEND N LINES
Insert a line for the indexed tool number after the active line. The function is only active if you are permitted to store multiple compensation data for a tool (MP7262 not equal to 0). The TNC inserts a copy of the tool data after the last available index and increases the index by 1. Application: e.g. stepped drill with more than one length compensation value.	INSERT LINE
Delete current line (tool).	DELETE LINE
Display / Do not display pocket numbers.	POCKET # DISPLAY HIDE
Display all tools / only those tools that are stored in the pocket table.	TOOLS DISPLAY HIDE

Leaving the tool table

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

Additional notes on tool tables

MP7266.x defines which data can be entered in the tool table and in which sequence the data is displayed.

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

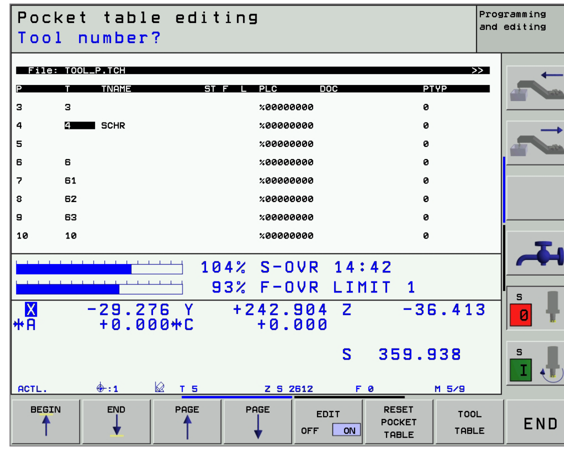
- The target file must exist.
- The file to be copied must contain only the columns (or lines) you want to replace.

To copy individual columns or lines, press the REPLACE FIELDS soft key (see “Copying a single file” on page 83).

Pocket table for tool changer

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

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



Editing a pocket table in a Program Run operating mode

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

PGM MGT

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

Abbr.	Input	Dialog
P	Pocket number of the tool in the tool magazine	–
T	Tool number	**Tool number?**
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 in column L (status L).	**Special tool?**
F	Fixed tool number. The tool is always returned to the same pocket in the tool magazine	**Fixed pocket? Yes = ENT / No = NO ENT**
L	Locked pocket (see also column ST)	**Pocket locked Yes = ENT / No = NO ENT**
PLC	Information on this tool pocket that is to be sent to the PLC	**PLC status?**
TNAME	Display of the tool name from TOOL.T	–
DOC	Display of the comment to the tool from TOOL.T	–

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 beginning of next line	NEXT LINE
Reset tool number column T	RESET COLUMN T

Calling tool data

To call a tool in the machining program, press the TOOL CALL key:

TOOL CALL

- **Tool number:** Enter the number or name of the tool. 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. If you wish to call a tool with other compensation values, also enter the index you defined in the tool table after the decimal point.
- **Spindle axis Z – XY plane**: Enter the tool axis. To transfer the default setting G17, press the ENT key. If you wish to select a different tool axis, use the soft keys.
- **Spindle speed S:** Enter the spindle speed directly or allow the TNC to calculate the spindle speed if you are working with cutting data tables. Press the S CALCULATE AUTOMAT. soft key. The TNC limits the spindle speed to the maximum value set in MP 3515. Confirm the speed you have entered by pressing the ENT key.
- **Feed rate F:** Enter the feed rate directly or allow the TNC to calculate the feed rate if you are working with cutting data tables. Press the F CALCULATE AUTOMAT. soft key. The TNC limits the feed rate to the maximum feed rate of the slowest axis (set in MP1010). F is effective until you program a new feed rate in a positioning block or a T block. Confirm the feed rate you have entered by pressing the ENT key.
- **Tool length oversize**: Enter the delta value for the tool length and confirm your entry with the ENT key.
- **Tool radius oversize**: Enter the delta value for the tool radius and confirm your entry with the ENT key.
- **Tool radius oversize 2**: Enter the delta value for the tool radius 2 and confirm with the ENT key.

Example: Tool call

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

```
N20 T 5.2 G17 S2500 DL+0.2 DR-1
```

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

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.

Tool change

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

Tool change position

The tool change position must be approachable without collision. With the miscellaneous functions **M91** and **M92,** you can enter machine-referenced (rather than workpiece-referenced) coordinates for the tool change position. If **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 "Interrupting machining," page 428).
- Change the tool.
- Resume program run (see "Resuming program run after an interruption," page 430).

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

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

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

The tool is 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 R0, RR, RL

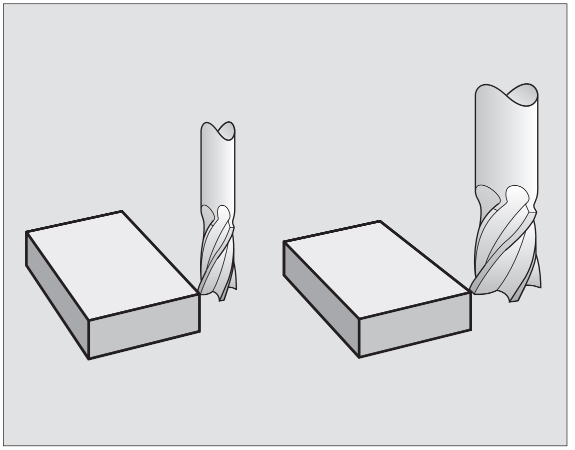
The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool.

5.3 Tool Compensation

Introduction

The TNC adjusts the spindle path in the tool axis by the compensation value for the tool length. In the working plane, it compensates the tool radius.

If you are writing the part program directly on the TNC, the tool radius compensation is effective only in the working plane. The TNC accounts for the compensation value in up to five axes including the rotary axes.



Tool length compensation

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

If you cancel a positive length compensation with **T0**, the distance between tool and workpiece will be reduced.

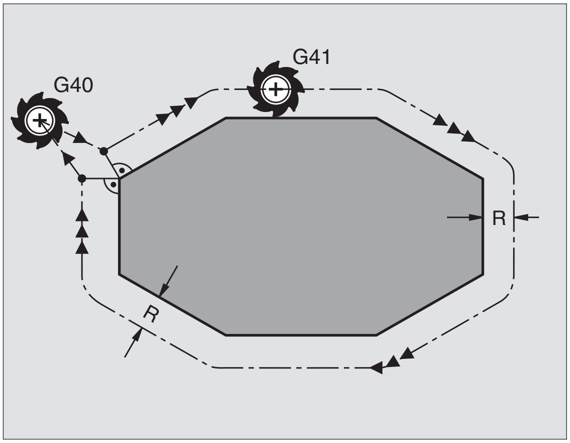
After a tool call, the path of the tool in the tool axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

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

Compensation value = $\mathbf{L} + \mathbf{DL}_T + \mathbf{DL}_{TAB}$, where

L:	is the tool length **L** from the **G99** block or tool table
DL $_{TL}$	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 program call with **%...,**
> - select a new program with PGM MGT.

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

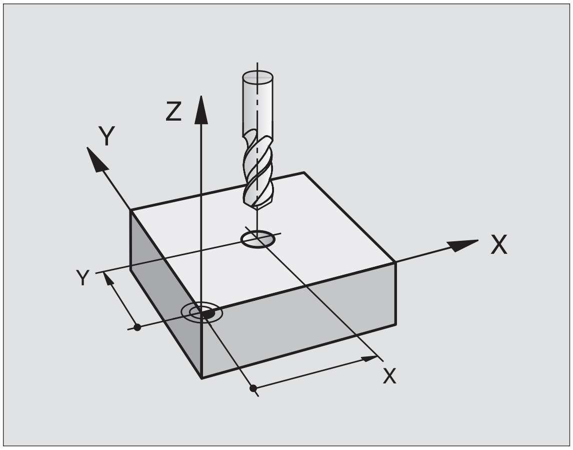
Compensation value = **R** + **DR**$_T$ + **DR**$_{TAB}$, where

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: R0

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

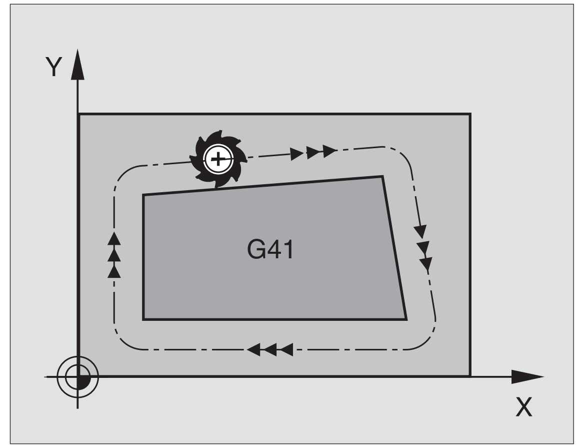
Applications: Drilling and boring, pre-positioning.

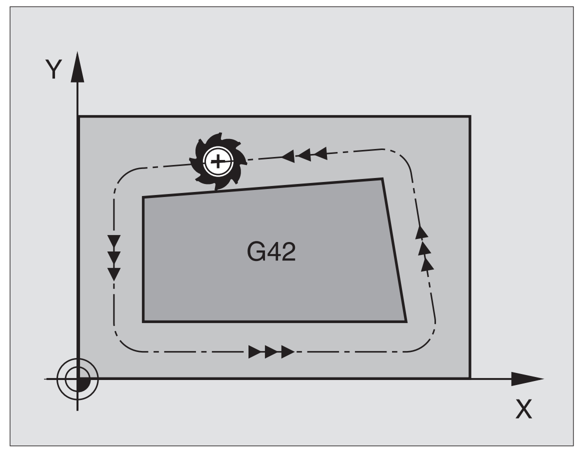


Contouring with radius compensation: G41 and G42

G42	The tool moves to the right of the programmed contour
G41	The tool moves to the left of the programmed contour

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





> Between two program blocks with different radius compensations (**G42** and **G41**) you must program at least one traversing block in the working plane without radius compensation (that is, with **G40**).
>
> Radius compensation does not take 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 as well in each following block, since otherwise the TNC will execute the radius compensation in the principal axis again.
>
> Whenever radius compensation is activated with **G42/G41** 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.

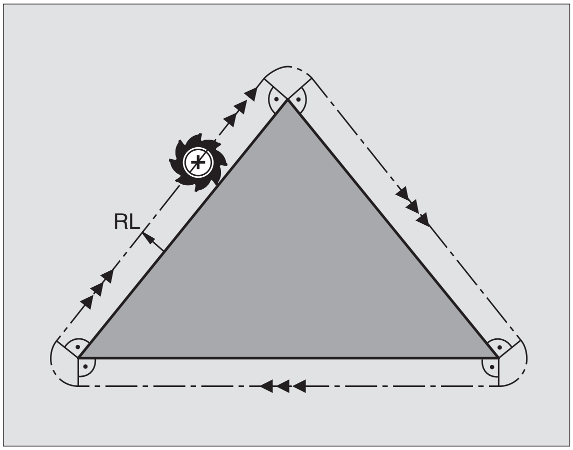
Entering radius compensation

Radius compensation is entered in a G01 block:

G41	To select tool movement to the left of the contour, select function G41, or
G42	To select tool movement to the right of the contour, select function G42, or
G40	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

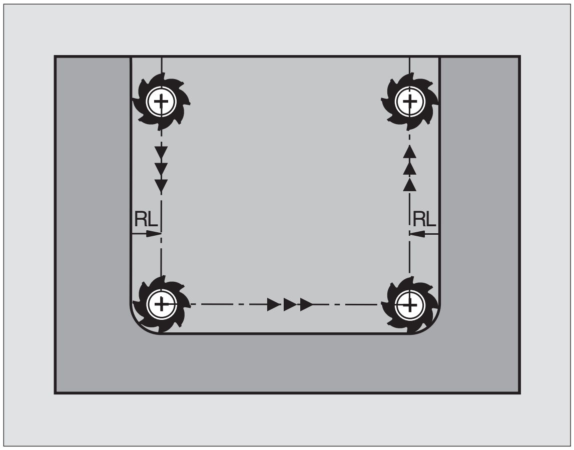
- Outside corners
  If you program radius compensation, the TNC moves the tool around outside corners either on a transitional arc or on a spline (selectable via MP7680). If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.
- Inside corners
  The TNC calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.



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

Machining corners without radius compensation

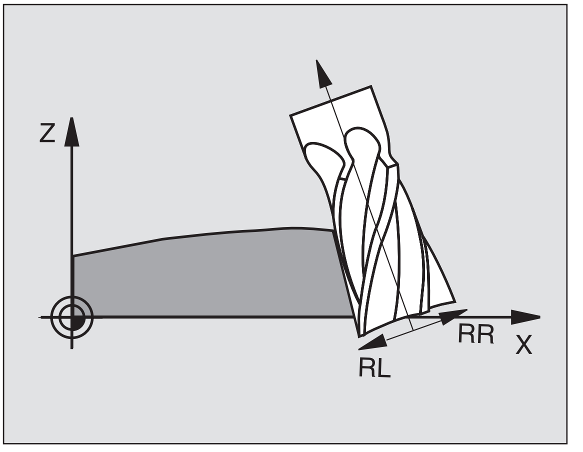
If you program the tool movement without radius compensation, you can change the tool path and feed rate at workpiece corners with the miscellaneous function **M90.** See “Smoothing corners: M90,” page 191.



5.4 Peripheral Milling: 3-D radius compensation with workpiece orientation

Function

With peripheral milling, the TNC displaces the tool perpendicular to the direction of movement and perpendicular to the tool direction by the sum of the delta values **DR** (tool table and **T** block). Determine the compensation direction with radius compensation **G41/G42** (see figure at upper right, traverse direction Y+).



For the TNC to be able to reach the set tool orientation, you need to activate the function **M128** (see "Maintaining the position of the tool tip when positioning with tilted axes (TCPM*): M128" on page 206) and subsequently the tool radius compensation. The TNC then positions the rotary axes automatically so that the tool can reach the orientation defined by the coordinates of the rotary axes with the active compensation.

This function is possible only on machines for which you can define spatial angles for the tilting axis configuration. Refer to your machine manual.

The TNC is not able to automatically position the rotary axes on all machines. Refer to your machine manual.

Danger of collision!

On machines whose rotary axes only allow limited traverse, sometimes automatic positioning can require the table to be rotated by 180°. In this case, make sure that the tool head does not collide with the workpiece or the clamps.

You can define the tool orientation in a G01 block as described below.

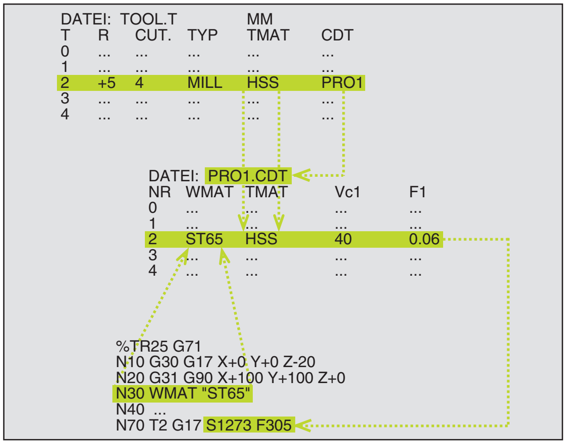
Example: Definition of the tool orientation with M128 and the coordinates of the rotary axes

`N10 G00 G90 X-20 Y+0 Z+0 B+0 C+0 *`	Pre-position
`N20 M128 *`	Activate M128
`N30 G01 G42 X+0 Y+0 Z+0 B+0 C+0 F1000 *`	Activate radius compensation
`N40 X+50 Y+0 Z+0 B-30 C+0 *`	Position rotary axis (tool orientation)

5.5 Working with Cutting Data Tables

Note

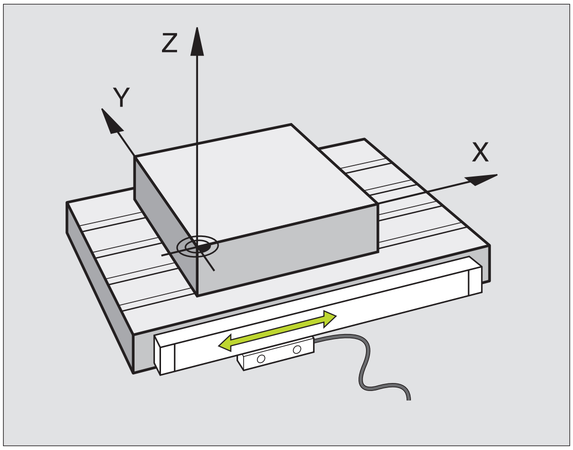
> The TNC must be specially prepared by the machine tool builder for the use of cutting data tables.
>
> Some functions or additional functions described here may not be provided on your machine tool. Refer to your machine manual.



Applications

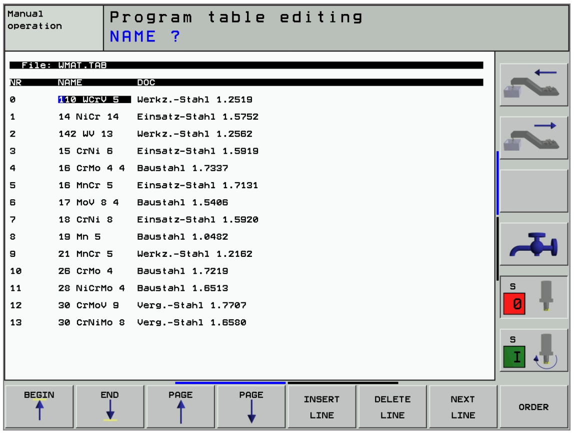
In cutting data tables containing various workpiece and cutting material combinations, the TNC can use the cutting speed V_C and the tooth feed f_Z to calculate the spindle speed S and the feed rate F. This calculation is only possible if you defined the workpiece material in the program and various tool-specific features in the tool table.

> Before you let the TNC automatically calculate the cutting data, the tool table from which the TNC is to take the tool-specific data must be first be activated in the Test Run mode (status S).



Editing function for cutting data tables	Soft key
Insert line	INSERT LINE
Delete line	DELETE LINE
Go to beginning of next line	NEXT LINE
Sort the table	ORDER N
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD
Edit the table format (2nd soft-key row)	EDIT FORMAT

Table for workpiece materials



Workpiece materials are defined in the table WMAT.TAB (see figure at upper right). WMAT.TAB is stored in the TNC:\ directory and can contain as many materials as you want. The name of the material type can have up to 32 characters (including spaces). The TNC displays the contents of the NAME column when you are defining the workpiece material in the program (see the following section).

> If you change the standard workpiece material table, you must copy it into a new directory. Otherwise your changes will be overwritten during a software update by the HEIDENHAIN standard data. Define the path in the TNC.SYS file with the code word WMAT= (see “Configuration file TNC.SYS,” page 153).
>
> To avoid losing data, save the WMAT.TAB file at regular intervals.

Defining the workpiece material in the NC program

In the NC program select the workpiece material from the WMAT.TAB table using the WMAT soft key:

WMAT

- Program the workpiece material: In the Programming and Editing operating mode, press the WMAT soft key.

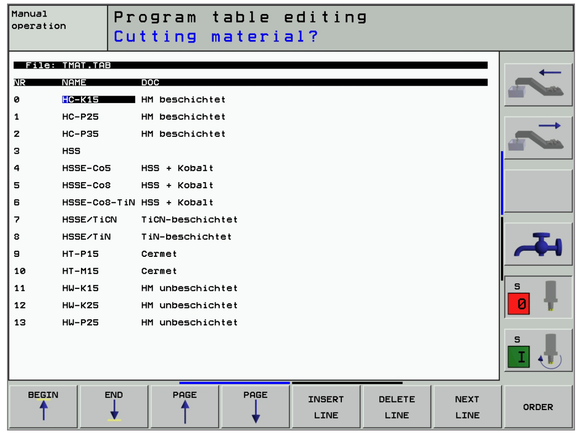
SELECTION WINDOW

- The WMAT.TAB table is superimposed: Press the SELECTION WINDOW soft key, and in a second window the TNC displays the list of materials that are stored in the WMAT.TAB table.
- Select your workpiece material by using the arrow keys to move the highlight onto the material you wish to select and confirming with the ENT key. The TNC transfers the selected material to the WMAT block.
- To terminate the dialog, press the END key.

> If you change the WMAT block in a program, the TNC outputs a warning. Check whether the cutting data stored in the T block are still valid.

Table for tool cutting materials

Tool cutting materials are defined in the TMAT.TAB table. TMAT.TAB is stored in the TNC:\ directory and can contain as many material names as you want (see figure at upper right). The name of the cutting material type can have up to 16 characters (including spaces). The TNC displays the NAME column when you are defining the tool cutting material in the TOOL.T tool table.

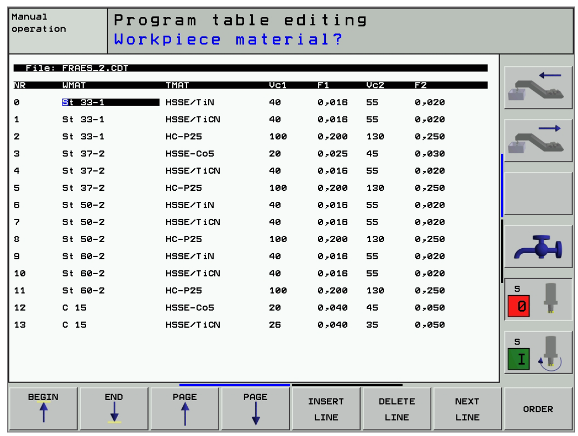


> If you change the standard tool cutting material table, you must copy it into a new directory. Otherwise your changes will be overwritten during a software update by the HEIDENHAIN standard data. Define the path in the TNC.SYS file with the code word TMAT= (see "Configuration file TNC.SYS," page 153).
>
> To avoid losing data, save the TMAT.TAB file at regular intervals.

Table for cutting data

Define the workpiece material / cutting material combinations with the corresponding cutting data in a file table with the file name extension .CDT; see figure at center right. You can freely configure the entries in the cutting data table. Besides the obligatory columns NR, WMAT and TMAT, the TNC can also manage up to four cutting speed (V_C) / feed rate (F) combinations.

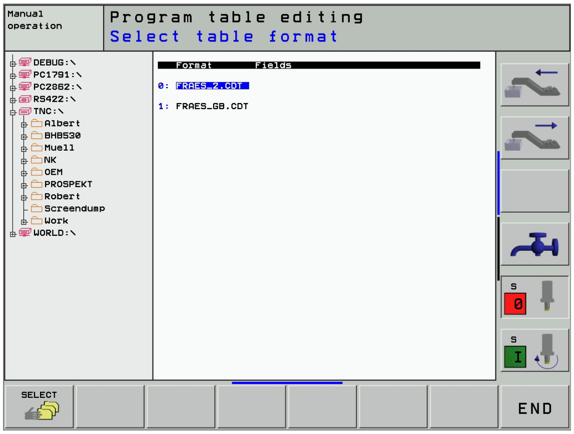
The standard cutting data table FRAES_2.CDT is stored in the directory TNC:\. You can edit FRAES_2.CDT, or add as many new cutting-data tables as you wish.



> If you change the standard cutting data table, you must copy it into a new directory. Otherwise your changes will be overwritten during a software update by the HEIDENHAIN standard data (see "Configuration file TNC.SYS," page 153).
>
> All of the cutting data tables must be stored in the same directory. If the directory is not the standard directory TNC:\, then behind the code word PCDT= you must enter the path in which your cutting data is stored.
>
> To avoid losing data, save your cutting data tables at regular intervals.

Creating a new cutting data table

- Select the Programming and Editing mode of operation.
- Press the PGM MGT key to select the file manager.
- Select the directory where the cutting data table is to be stored.
- Enter any file name with file name extension .CDT, and confirm with ENT.
- On the right half of the screen, the TNC displays various table formats (machine-dependent, see example in figure at right). These tables differ from each other in the number of cutting speed / feed rate combinations they allow. Use the arrow keys to move the highlight onto the table format you wish to select and confirm with ENT. The TNC generates a new, empty cutting data table.



Data required for the tool table

- Tool radius—column R (DR)
- Number of teeth (only with tools for milling)—column CUT
- Tool type—column TYPE
- The tool type influences the calculation of the feed rate:
  Milling tool: $F = S \cdot f_Z \cdot z$
  All other tools: $F = S \cdot f_U$
  S: Spindle speed
  f_Z: Feed per tooth
  f_U: Feed per revolution
  z: Number of teeth
- Tool cutting material—column TMAT
- Name of the cutting data table for which this tool will be used—column CDT
- In the tool table, select the tool type, tool cutting material and the name of the cutting data table via soft key (see “Tool table: Tool data for automatic speed/feed rate calculation.,” page 135).

Working with automatic speed / feed rate calculation

1 If it has not already been entered, enter the type of workpiece material in the file WMAT.TAB.

2 If it has not already been entered, enter the type of cutting material in the file TMAT.TAB.

3 If not already entered, enter all of the required tool-specific data in the tool table:
- Tool radius
- Number of teeth
- Tool type
- Tool material
- The cutting data table for each tool

4 If not already entered, enter the cutting data in any cutting data table (CDT file).

5 Test Run operating mode: Activate the tool table from which the TNC is to take the tool-specific data (status S).

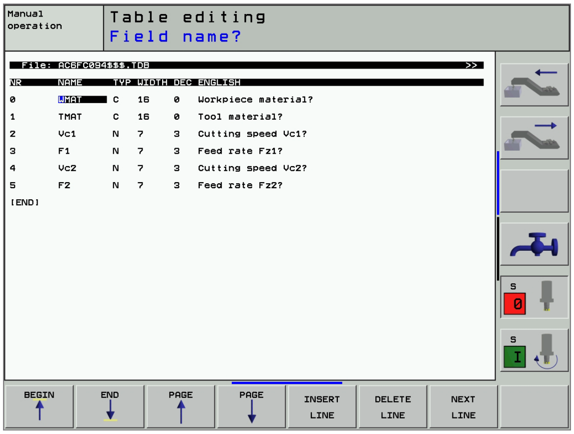
6 In the NC program, set the workpiece material by pressing the WMAT soft key.

7 In the NC program, let the TOOL CALL block automatically calculate spindle speed and feed rate via soft key.

Changing the table structure

Cutting data tables constitute so-called "freely-definable tables" for the TNC. You can change the format of freely definable tables by using the structure editor.

> The TNC can process up to 200 characters per line and up to 30 columns.
>
> If you insert an additional column into an existing table, the TNC does not automatically shift the values that have been entered.



Calling the structure editor

Press the EDIT FORMAT soft key (2nd soft-key level). The TNC opens the editing window (see figure at right), in which the table structure is shown "rotated by 90°." In other words, a line in the editing window defines a column in the associated table. The meanings of the structure commands (header entries) are shown in the table at right.

Exiting the structure editor

Press the END key. The TNC changes data that was already in the table into the new format. Elements that the TNC could not convert into the new structure are indicated with a hash mark # (e.g., if you have narrowed the column width).

Structure command	Meaning
NR	Column number
NAME	Overview of columns
TYPE	N: Numerical input C: Alphanumeric input
WIDTH	Width of column. For type N including algebraic sign, comma, and decimal places.
DEC	Number of decimal places (max. 4, effective only for type N)
ENGLISH to HUNGARIAN	Language-dependent dialogs (max. 32 characters)

Data transfer from cutting data tables

If you output a file type .TAB or .CDT via an external data interface, the TNC also transfers the structural definition of the table. The structural definition begins with the line #STRUCTBEGIN and ends with the line #STRUCTEND. The meanings of the individual code words are shown in the table "Structure Command" (see "Changing the table structure," page 151). Behind #STRUCTEND the TNC saves the actual content of the table.

Configuration file TNC.SYS

You must use the configuration file TNC.SYS if your cutting data tables are not stored in the standard directory TNC:\. In TNC.SYS you must then define the paths in which you have stored your cutting data tables.

The TNC.SYS file must be stored in the root directory TNC:\.

Entries in TNC.SYS	Meaning
WMAT=	Path for workpiece material table
TMAT=	Path for cutting material table
PCDT=	Path for cutting data tables

Example of TNC.SYS

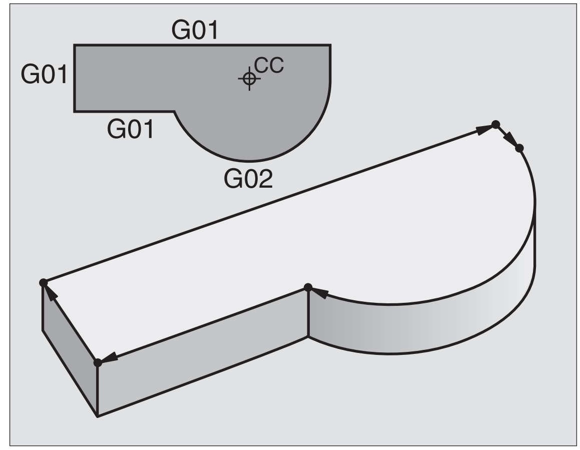
```
WMAT=TNC:\CUTTAB\WMAT_GB.TAB
TMAT=TNC:\CUTTAB\TMAT_GB.TAB
PCDT=TNC:\CUTTAB\
```

6

**Programming:
Programming Contours**

6.1 Tool Movements

Path functions



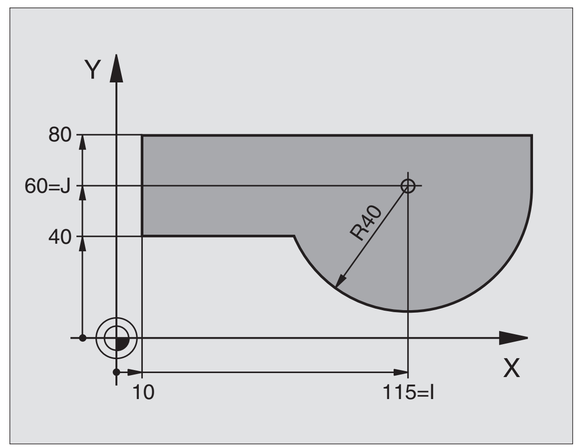
A workpiece contour is usually composed of several contour elements, such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs.**

Miscellaneous functions M

With the miscellaneous functions of the TNC you can control:

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

Subprograms and Program Section Repeats



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

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

Programming with Q parameters

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

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 usually do this by entering **the coordinates of the end points of the contour elements** given in the production drawing. The TNC calculates the actual path of the tool from these coordinates, using the tool data and radius compensation.

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

Movement parallel to the machine axes

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

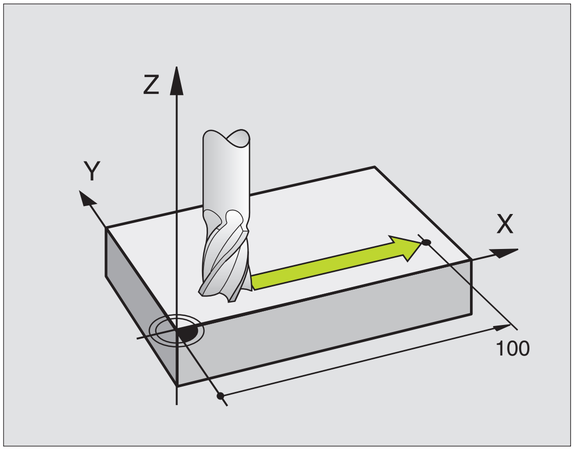
Depending on the machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

```
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).



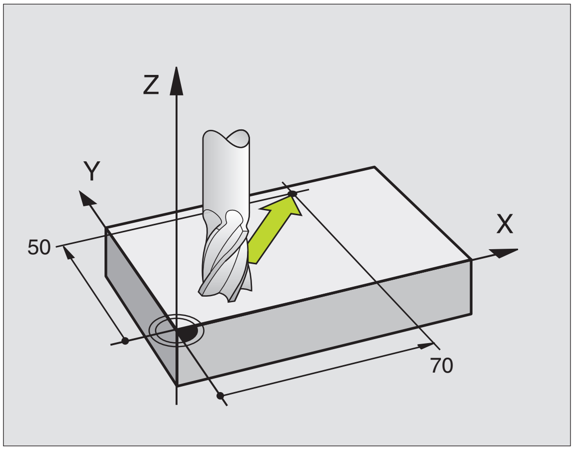
Movement in the main planes

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).

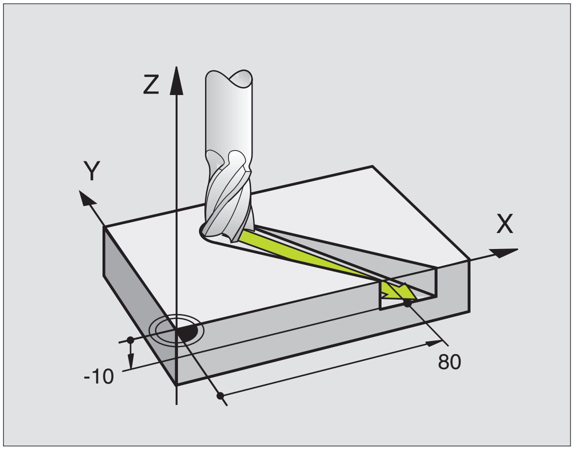


Three-dimensional movement

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 *
```



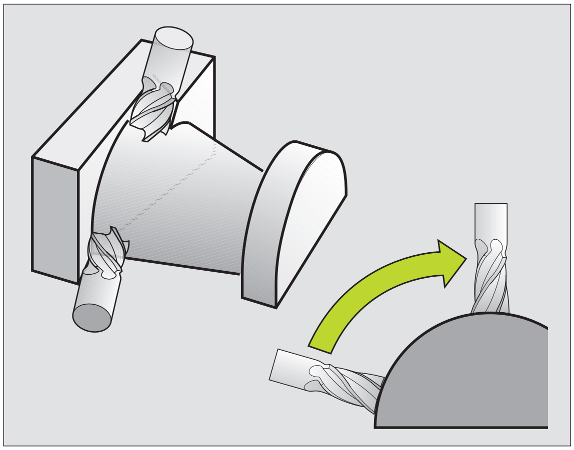
Entering more than three coordinates

The TNC can control up to 5 axes simultaneously. Machining with 5 axes, for example, moves 3 linear and 2 rotary axes simultaneously.

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

Example:

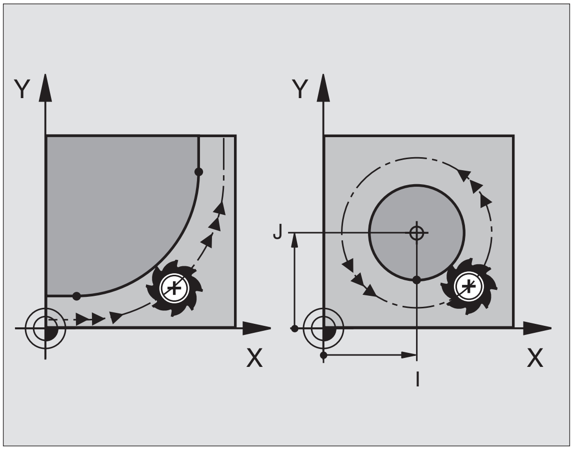
```
N G01 G40 X+20 Y+10 Z+2 A+15 C+6 F100 M3 *
```



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:

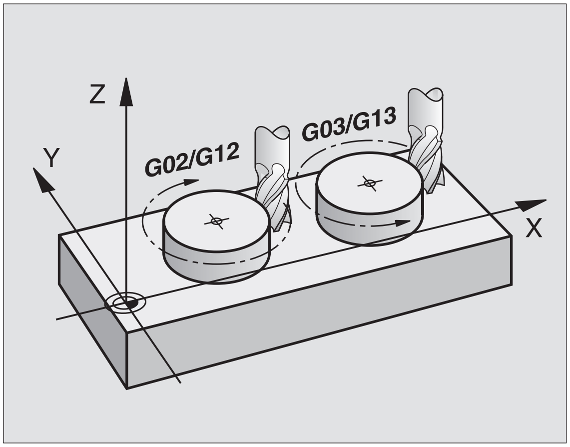
Spindle axis	Main plane	Circle center
Z (G17)	**XY,** also UV, XV, UY	**I, J**
Y (G18)	**ZX,** also WU, ZU, WX	**K, I**
X (G19)	**YZ,** also VW, YW, VZ	**J, K**

You can program circles that do not lie parallel to a main plane by using the function for tilting the working plane (see "WORKING PLANE (Cycle G80)," page 359) or Q parameters (see "Principle and Overview," page 386).

Direction of rotation for circular movements

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

- Clockwise direction of rotation: G02/G12
- Counterclockwise direction of rotation: G03/G13



Radius compensation

The radius compensation must be in the block in which you move to the first contour element. You cannot begin radius compensation in a circle block. It must be activated beforehand in a straight-line block (see “Path Contours—Cartesian Coordinates,” page 164).

Pre-positioning

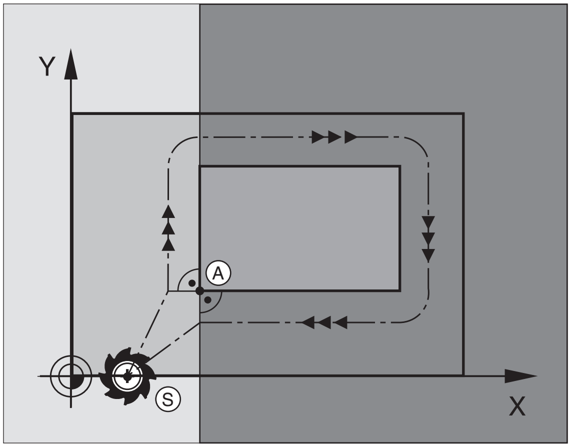
Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece.

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

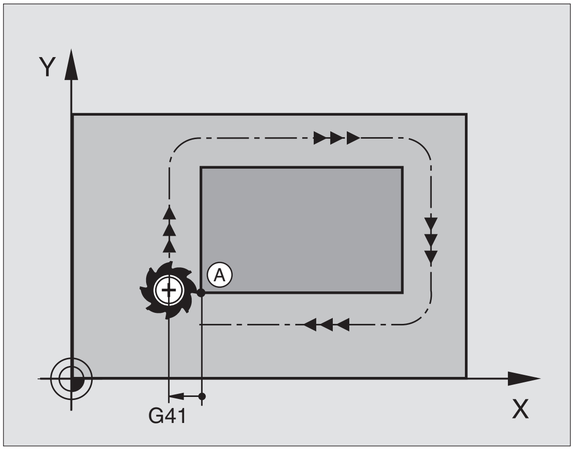


Example

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.

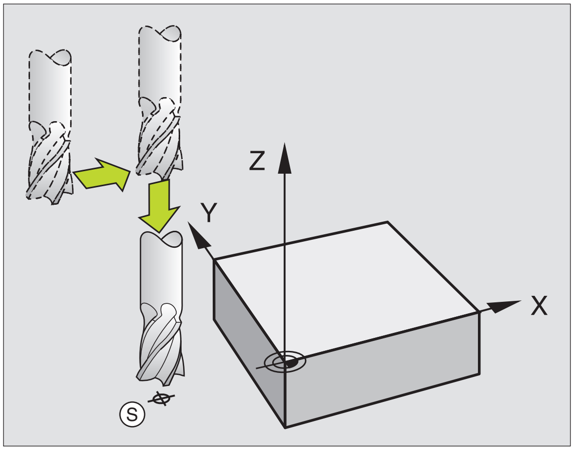


Approaching the starting point in the spindle axis

When the starting point is approached, 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

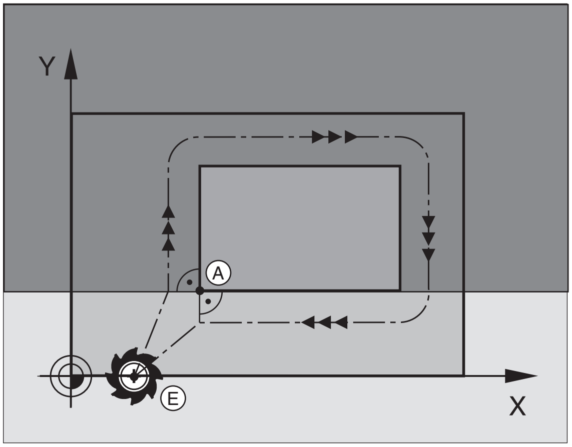
```
N30 G00 G40 X+20 Y+30 *
N40 Z-10 *
```



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.

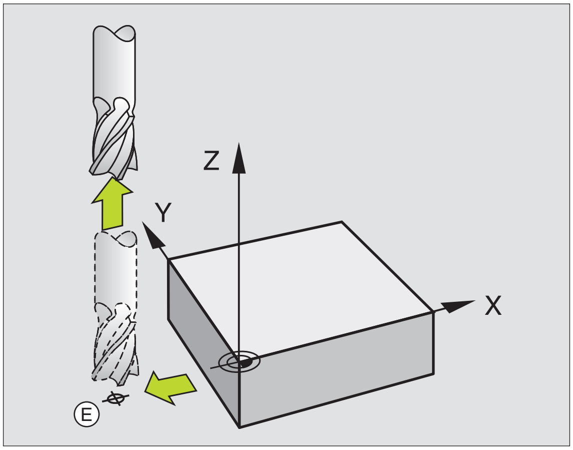


Example

Figure at upper 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 center right.



Example NC blocks

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

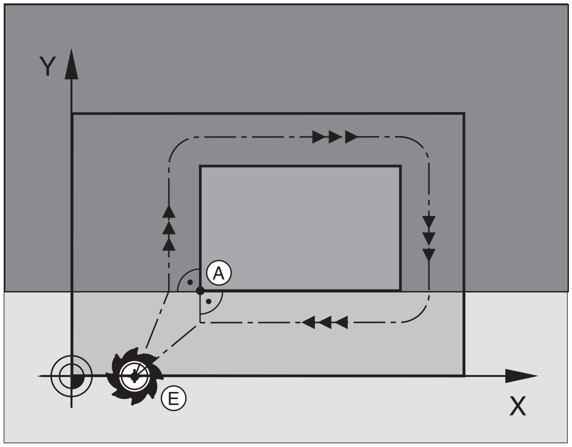
Common starting and end points

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.

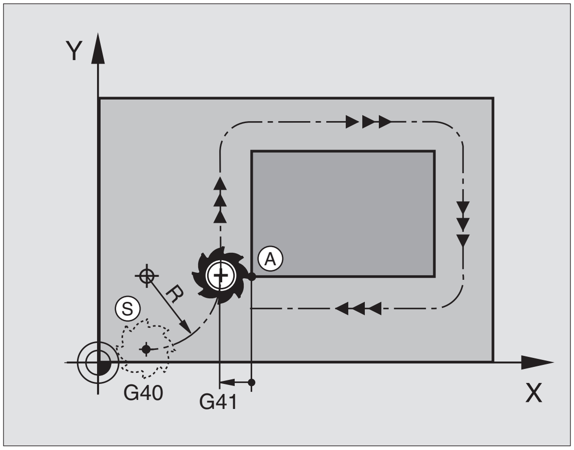
Example

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

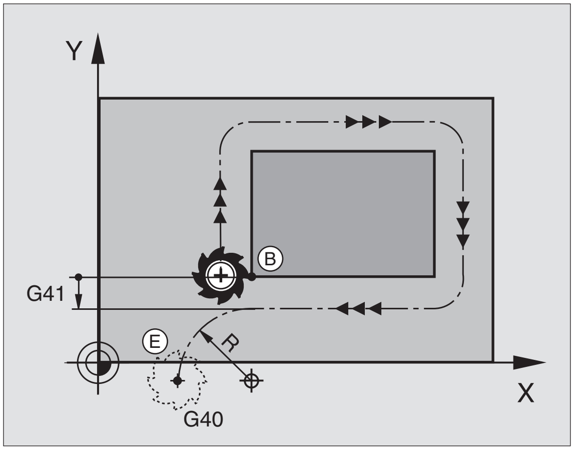
The starting point and the end point lie outside the workpiece, close to the first and last contour points. They are to be programmed without radius compensation.

Approach

- **G26** is entered after the block in which the first contour element is programmed: This will be the first block with radius compensation **G41/G42.**

Departure

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



> The radius for **G26** and **G27** must be selected so that the TNC can 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	
. . .	Last contour point
N210 G27 R5 *	Tangential departure with radius R = 5 mm
N220 G00 G40 X-30 Y+50 *	End point

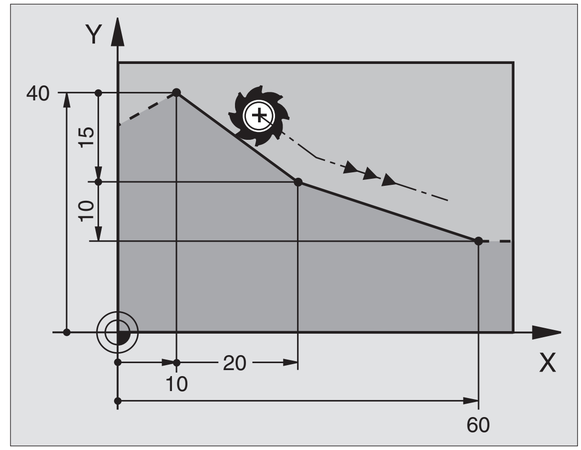
6.4 Path Contours—Cartesian Coordinates

Overview of path functions

Tool movement	Function	Required input
Straight line at feed rate Straight line at rapid traverse	**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 arc with tangential connection to the preceding contour element	**G06**	Coordinates of the arc end point
Circular arc with tangential connection to the preceding and subsequent contour elements	**G25**	Rounding-off radius **R**

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

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



Programming

G 1

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

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 *
```

Actual position capture

With the actual-position-capture function, you can transfer any desired axis position into a block:

- In the Manual Operation mode, move the tool to the position you wish to capture.
- Switch the screen display to Programming and Editing.
- Select the program block into which you want to take over an axis position.

- Select the actual position capture function: In the soft-key row the TNC displays the axes whose positions can be transferred.

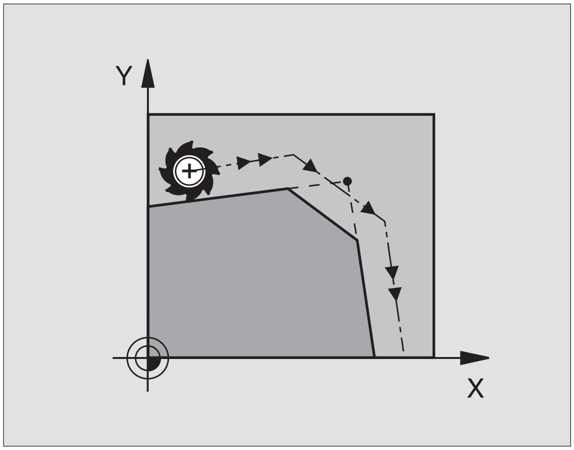
AXIS X

- Select the axis, e.g. X: The TNC writes the current position of the selected axis into the active input box.

Inserting a chamfer CHF between two straight lines

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

- The blocks before and after the **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.



Programming

G 24

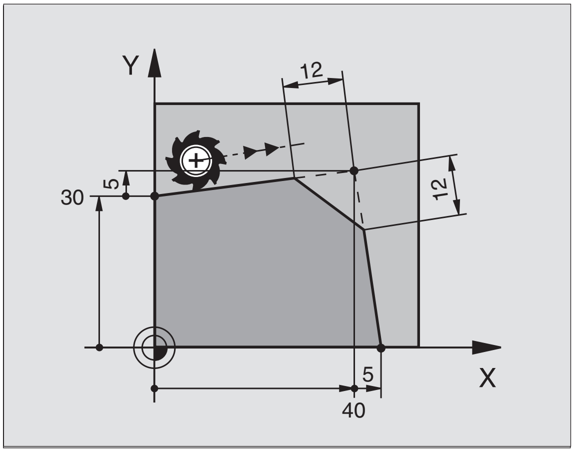
▶ **Chamfer side length:** Length of the chamfer

Further entries, if necessary:

▶ **Feed rate F** (only effective in **G24** block)

Example NC blocks

```
N70 G01 G41 X+0 Y+30 F300 M3 *
N80 X+40 G91 Y+5 *
N90 G24 R12 F250 *
N100 G91 X+5 G90 Y+0 *
```



You cannot start a contour with a **G24** block.

A chamfer is possible only in the working plane.

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

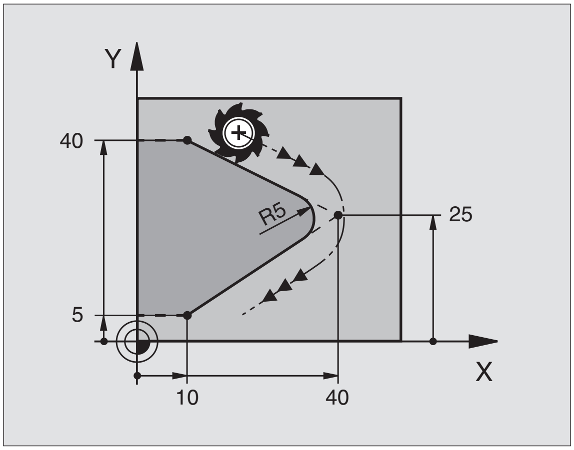
A feed rate programmed in the **G24** block is effective only in that block. After the **G24** block, the previous feed rate becomes effective again.

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.



Programming

G 25

- **Rounding-off radius:** Enter the radius

Further entries, if necessary:
- **Feed rate F** (only effective in **G25** block)

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.

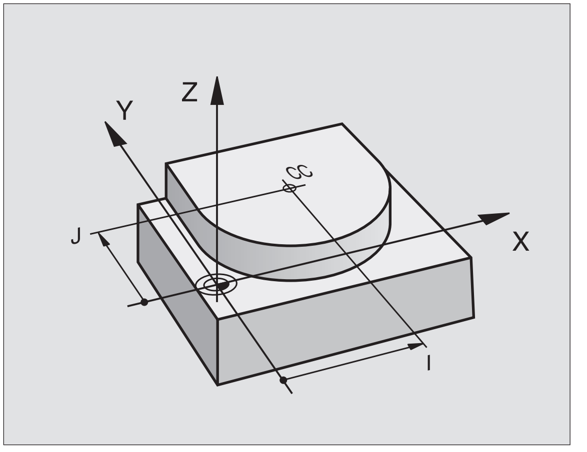
A feed rate programmed in the **G25** block is effective only in that block. After the **G25** block, the previous feed rate becomes effective again.

You can also use a **G25** block for a tangential contour approach (see "Tangential approach and departure," page 162).

Circle center I, J

You can define a circle center for circles that are programmed with the functions G02, G03 or G05. This is done in the following ways:

- Entering the Cartesian coordinates of the circle center, or
- Using the last programmed circle center (G29),
- Transferring the coordinates with the actual-position-capture function.



Programming

I J

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

Example NC blocks

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

or

```
N10 G00 G40 X+25 Y+25 *
N20 G29 *
```

The program blocks N10 and N20 do not refer to the illustration.

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.

Entering 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.

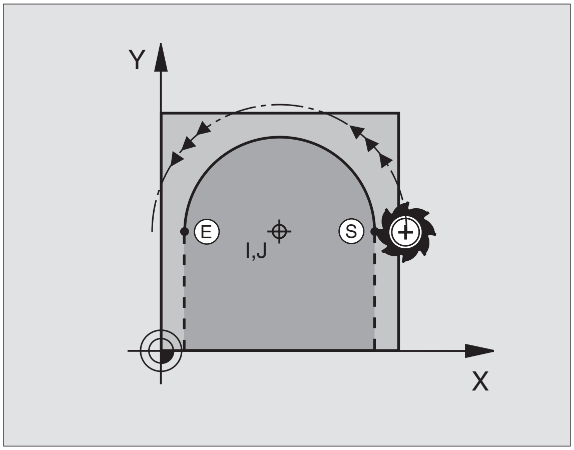
The only effect of **I** and **J** is to define a position as a circle center—the tool does not move to the position.

The circle center is also the pole for polar coordinates.

If you wish to define the pole in parallel axes, first press the **I** (**J**) key on the ASCII keyboard, and then the orange axis key for the corresponding parallel axis.

Circular path G02/G03/G05 around 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 programmed direction: **G05.** The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

▶ 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

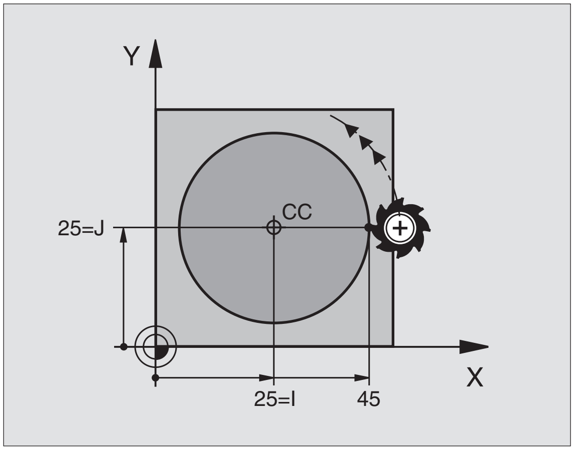
▶ Miscellaneous function M

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

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

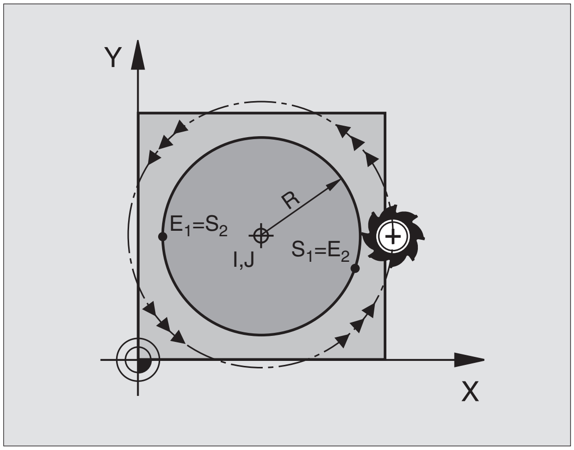


The starting and end points of the arc must lie on the circle.

Input tolerance: up to 0.016 mm (selected with MP7431).

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 programmed direction: **G05.** The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

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 CR blocks in succession:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first.

Central angle CCA and arc radius R

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

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

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

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

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

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

Example NC blocks

```
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)
```

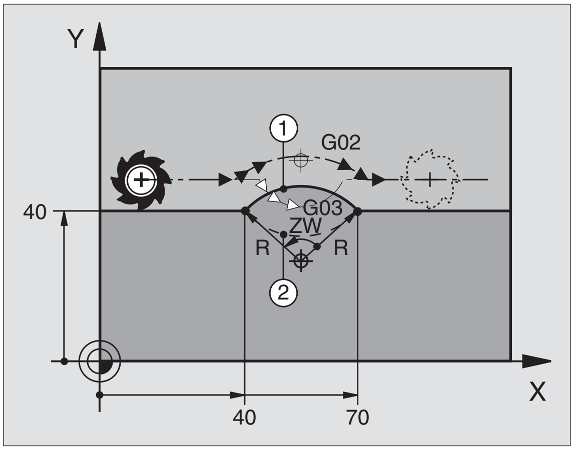
or

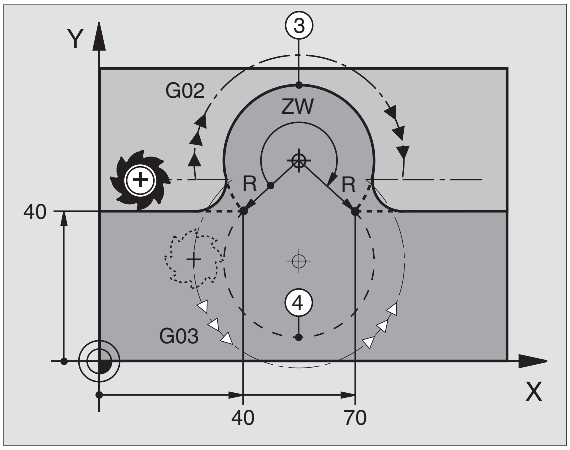
```
N110 G03 X+70 Y+40 R-20 * (ARC 4)
```

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 99.9999 m.

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



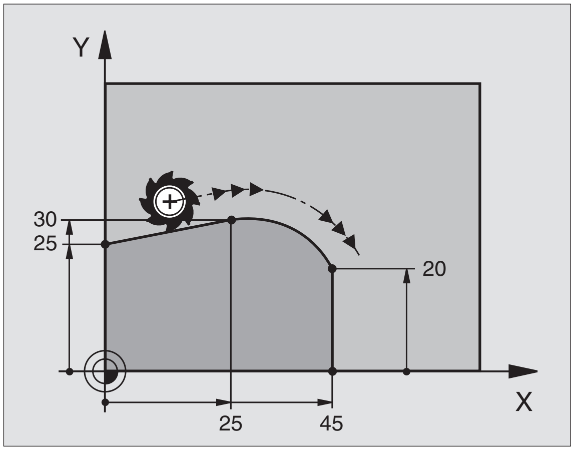


Circular path G06 with tangential approach

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

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

The contour element to which the tangential arc connects must be programmed immediately before the **G06** block. This requires at least two positioning blocks.



Programming

G 6

▶ Enter the coordinates of the arc end point.

Further entries, if necessary:

▶ Feed rate F

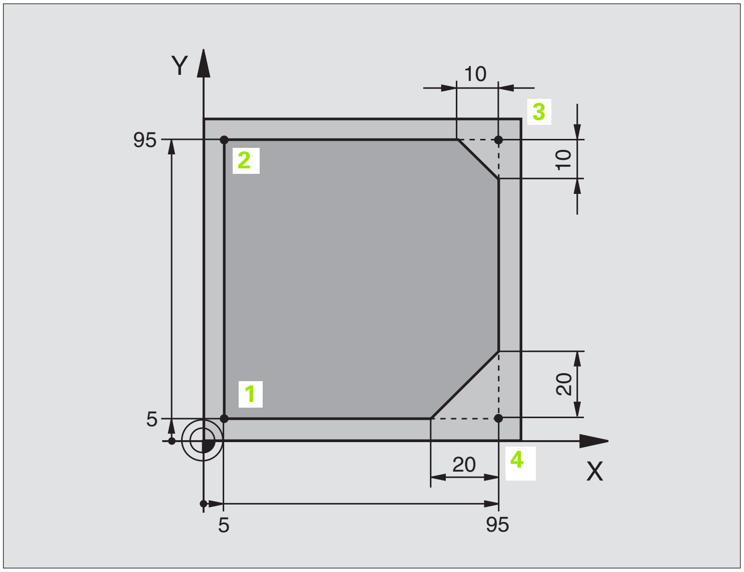
▶ Miscellaneous function M

Example NC blocks

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

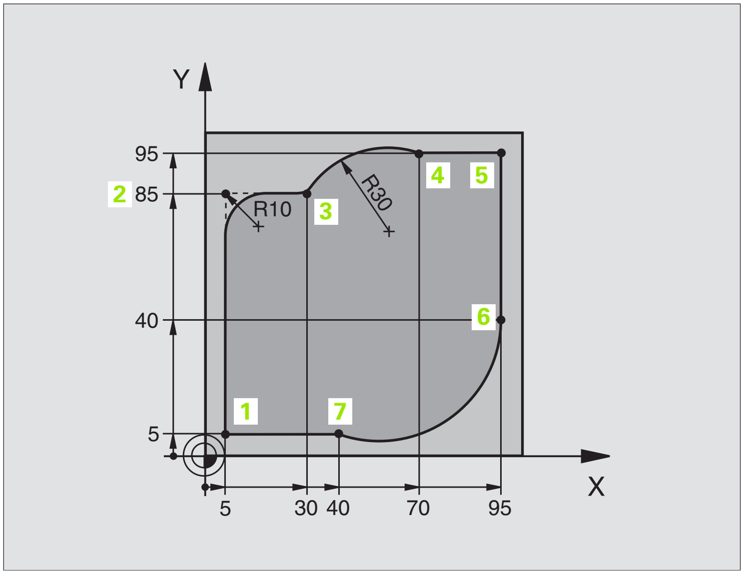
A tangential arc is a two-dimensional operation: the coordinates in the **G06** block and in the contour element preceding it must be in the same plane of the arc.

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 in the tool axis, end program
N999999 %LINEAR G71 *	

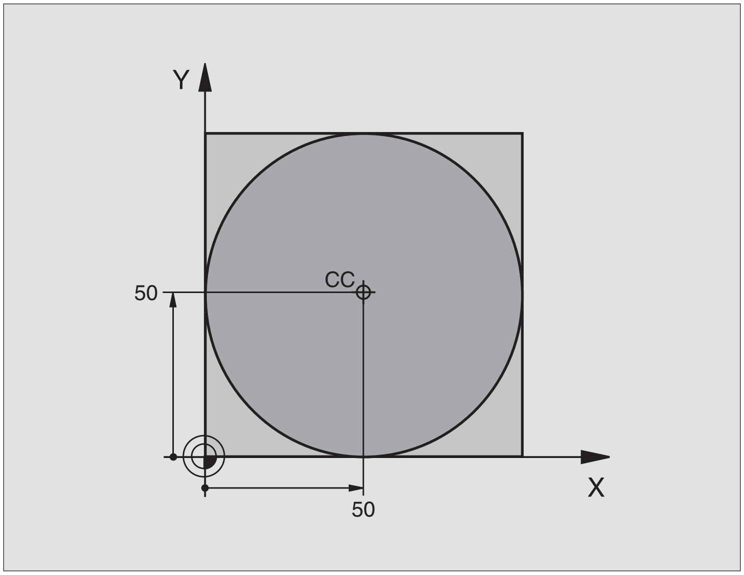
Example: Circular movements with Cartesian coordinates



`%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, feed rate: 150 mm/min
`N120 X+30 *`	Move to point 3: Starting point of the arc
`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 *	Depart the contour on a circular arc with tangential connection
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 tool 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 *`	Tool call
`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 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 tool axis, end of program
`N999999 %C-CC G71 *`	

6.5 Path Contours—Polar Coordinates

Overview of path functions with polar coordinates

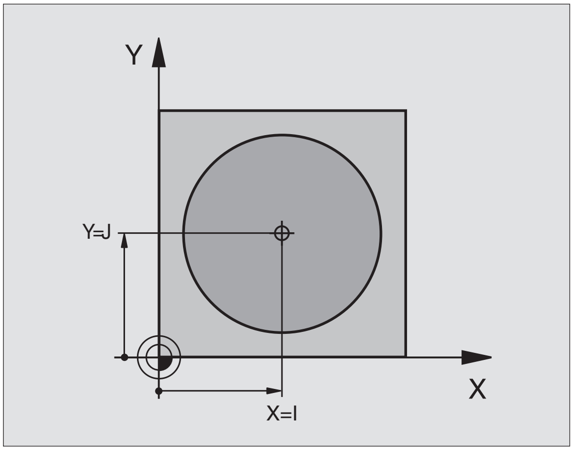
With polar coordinates you can define a position in terms of its angle **H** and its distance **R** relative to a previously defined pole **I, J** (see "Definition of pole and angle reference axis," page 66).

Polar coordinates are useful with:

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

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

Programming

I J

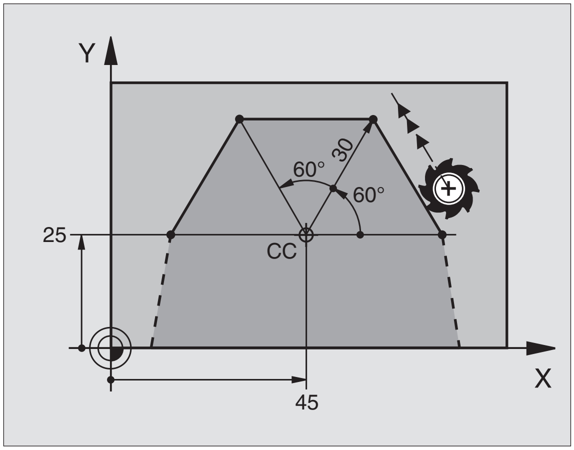
▶ Enter Cartesian coordinates for the pole, or if you want to use the last programmed position, enter **G29.** Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole.

Example NC blocks

```
N120 I+45 J+45 *
```

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.



Programming

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:

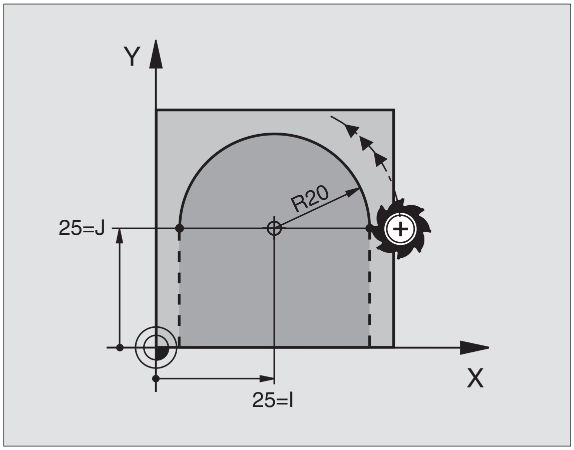
- Angle from angle reference axis to **R** is counterclockwise: **H** >0
- Angle from angle reference axis to **R** is clockwise: **H** <0

Example NC blocks

```
N120 I+45 J+45 *
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 programmed direction: **G15.** The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

G 13

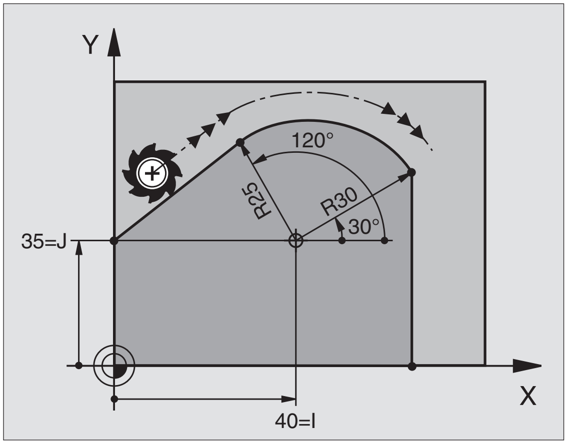
- Polar-coordinates angle **H:** Angular position of the arc end point between -5400° and +5400°

Example NC blocks

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

Circular arc with tangential connection

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



Programming

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

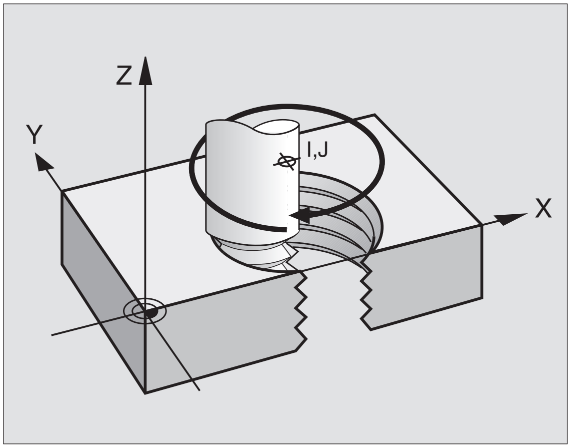
```
N120 I+40 J+35 *
N130 G01 G42 X+0 Y+35 F250 M3 *
N140 G11 R+25 H+120 *
N150 G16 R+30 H+30 *
N160 G01 Y+0 *
```

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

Helical interpolation

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

A helix is programmed only in polar coordinates.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

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

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

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

Shape of the helix

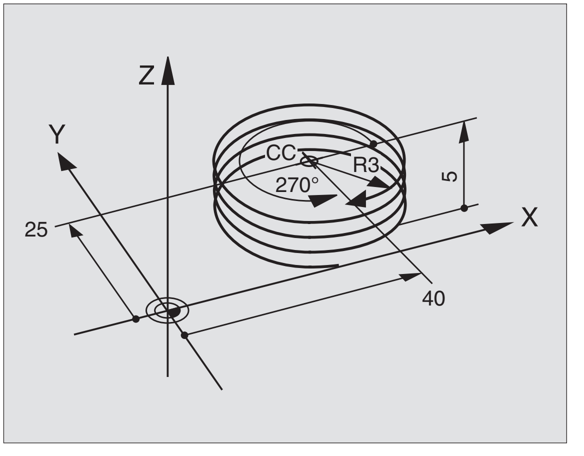
The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction	Radius comp.
Right-handed	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 more than 15 revolutions, program the helix in a program section repeat (see "Program Section Repeats," page 374)



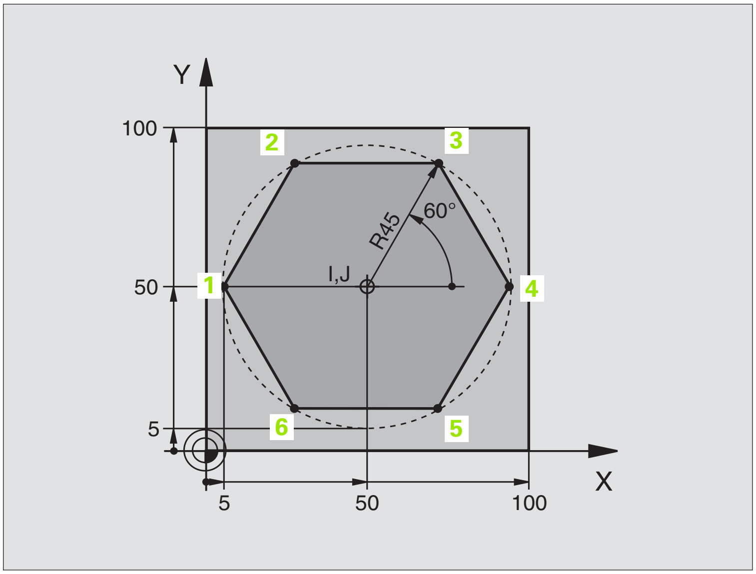
G 12

- Polar coordinates angle H: Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, specify the tool axis with an axis selection key.**
- Enter the coordinate for the height of the helix in incremental dimensions.
- Enter the radius compensation **G41/G42** according to the table above.

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

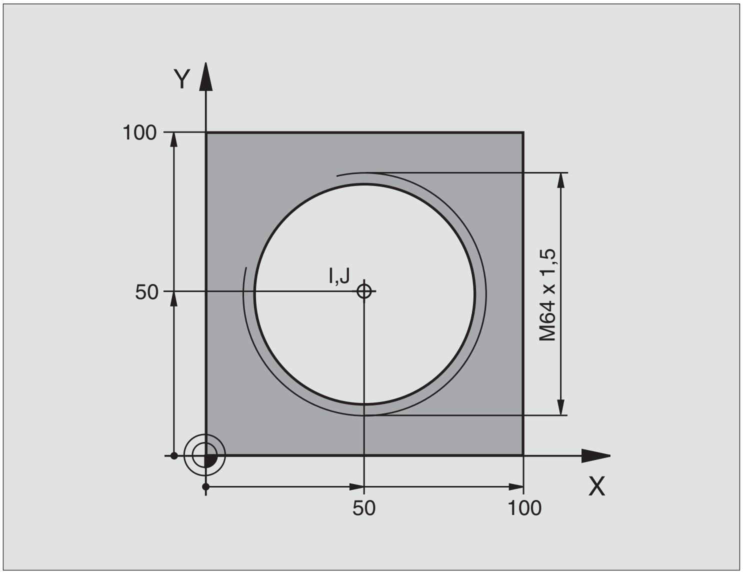
```
N120 I+40 J+25 *
N130 G01 Z+0 F100 M3 *
N140 G11 G41 R+3 H+270 *
N150 G12 G91 H-1800 Z+5 *
```

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 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Define the datum for polar coordinates
`N60 I+50 J+50 *`	Retract the tool
`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 *`	Approach the contour at point 1
`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 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 *	Tool call
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 *	connection
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 in the tool axis, end program
N180 G00 Z+250 M2 *	

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 *	Tangential approach

`N110 G98 L1 *`	Identify beginning of program section repeat
`N120 G13 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

Fundamentals

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

- 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. Refer to your machine manual.

You can enter up to two M functions at the end of a positioning block.

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

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

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

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

Overview

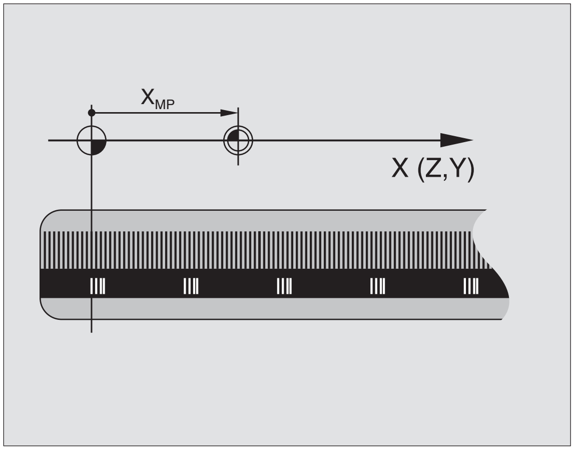
M	Effect	Effective at block start	end
M00	Stop program run Spindle STOP Coolant OFF		■
M01	Optional program STOP		■
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (depends on MP7300)		■
M03	Spindle ON clockwise	■	
M04	Spindle ON counterclockwise	■	
M05	Spindle STOP		■
M06	Tool change Spindle STOP Program run stop (depends on MP7440)		■
M08	Coolant ON	■	
M09	Coolant OFF		■
M13	Spindle ON clockwise Coolant ON	■	
M14	Spindle ON counterclockwise Coolant ON	■	
M30	Same as M02		■

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 (Without a 3-D Touch Probe)," page 50).

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 shown with respect to the machine datum. Switch the display of coordinates in the status display to REF (see "Status Displays," page 37).

Behavior with M92—Additional machine datum

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

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

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

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

Effect

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

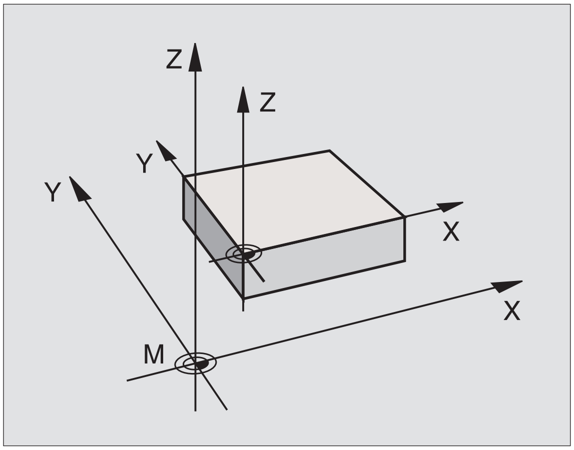
M91 and M92 take effect at the start of block.

Workpiece datum

If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes (see "General User Parameters" on page 466).

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.



M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the set datum (see "Showing the Workpiece in the Working Space," page 453).

Activating the most recently entered datum: M104

Function

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 a non-tilted coordinate system with a tilted working plane: M130

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.

> Subsequent positioning blocks or fixed cycles are carried out in a tilted coordinate system. This can lead to problems in fixed cycles with absolute pre-positioning.
>
> The function M130 is allowed only if the tilted working plane function is active.

Effect

M130 functions only in straight-line blocks without tool radius compensation and in blocks in which M130 is programmed.

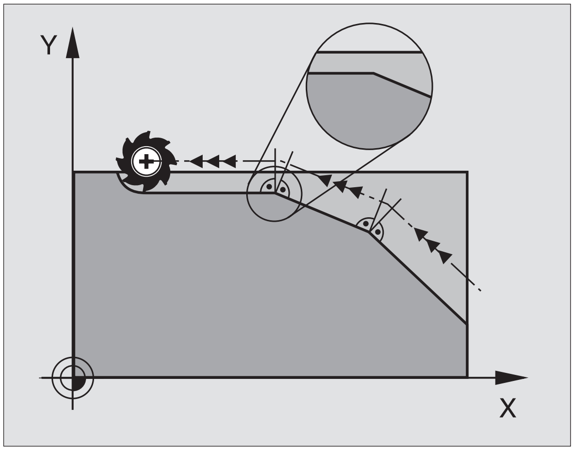
7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

Standard behavior

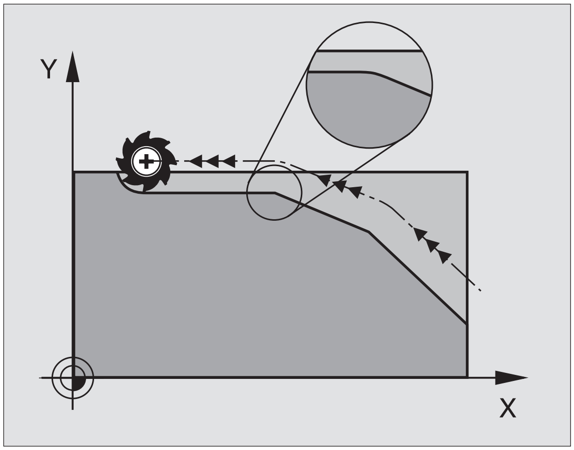
The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an exact stop.

In program blocks with radius compensation (**G41/G42**), 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.



Application example: 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.

Insert rounding arc between straight lines: M112

Compatibility

For reasons of compatibility, the M112 function is still available on the iTNC 530 control. However, to define the tolerance for fast contour milling, HEIDENHAIN recommends the use of the TOLERANCE cycle for these TNCs (see "TOLERANCE (Cycle G62)," page 369).

Do not include points when executing non-compensated line blocks: M124

Standard behavior

The TNC runs all line blocks that have been entered in the active program.

Behavior with M124

When running **non-compensated line blocks** with very small point intervals, you can use parameter **E** to define a minimum point interval up to which the TNC will not include points during execution.

Effect

M124 becomes effective at the start of block.

The TNC automatically resets M124 if you select a new program.

Programming M124

If you enter M124 in a positioning block, the TNC continues the dialog for this block by asking you the minimum distance between points **E.**

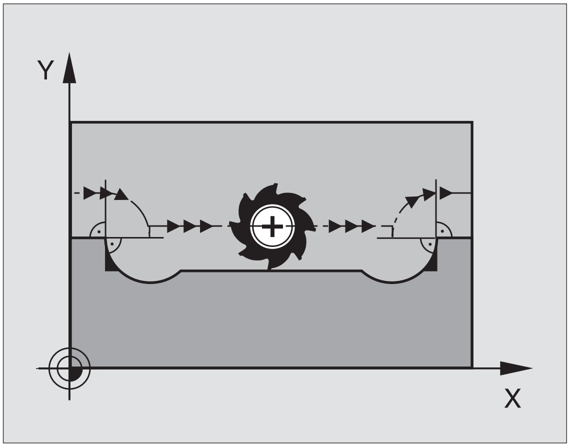
You can also define **E** through Q parameters (see "Programming: Q Parameters" on page 385).

Machining small contour steps: M97

Standard behavior

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

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



Behavior with M97

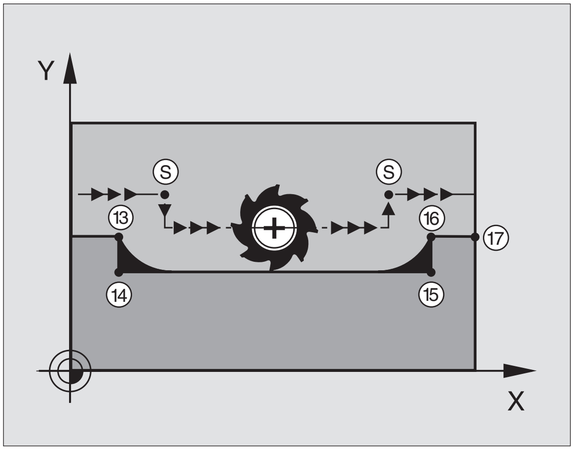
The TNC calculates the intersection of the contour elements—as at inside corners—and moves the tool over this point.

Program M97 in the same block as the outside corner.

Effect

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

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



Example NC blocks

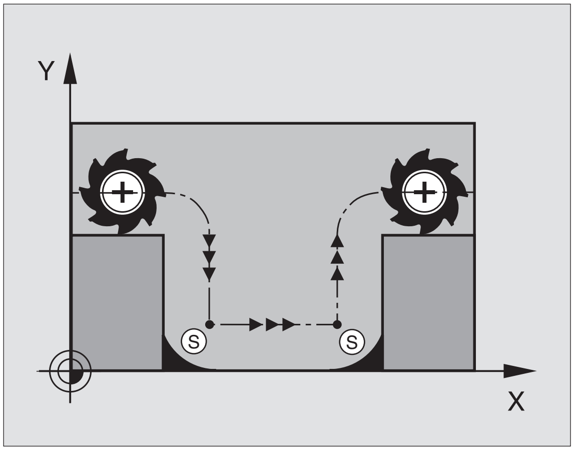
`N50 G99 G01 ... R+20 *`	Large tool radius
`...`	
`N130 X ... Y ... F .. M97 *`	Move to contour point 13
`N140 G91 Y-0.5 .... F.. *`	Machine small contour step 13 to 14
`N150 X+100 ... *`	Move to contour point 15
`N160 Y+0.5 ... F.. M97 *`	Machine small contour step 15 to 16
`N170 G90 X ... Y ... *`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



Behavior with M98

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

Effect

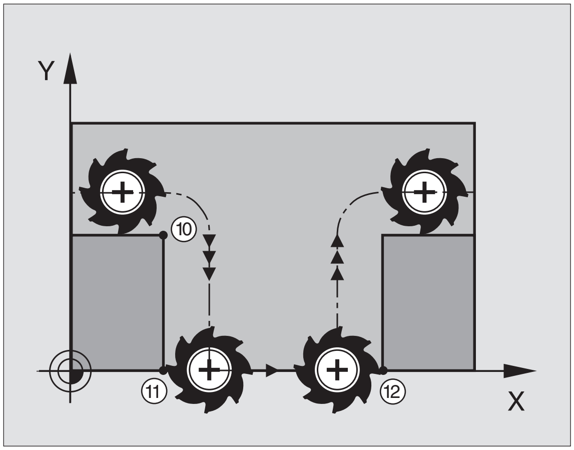
M98 is effective only in the blocks in which it is programmed.

M98 takes effect at the end of block.

Example NC blocks

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

```
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):
`N107 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

Feed rate in millimeters per spindle revolution: M136

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 millimeters 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 so that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110

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

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

Effect

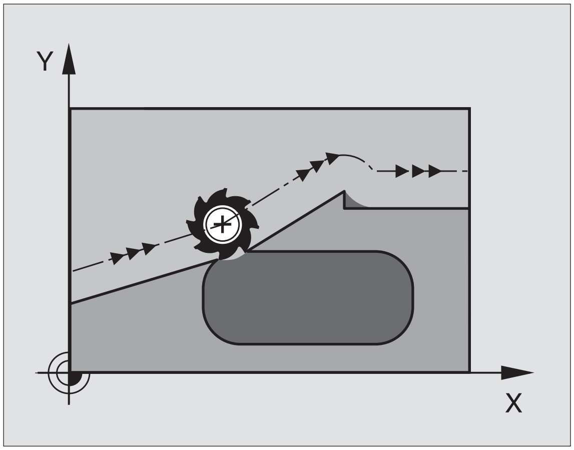
M109 and M110 become effective at the start of block.
To cancel M109 and M110, enter M111.

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

Standard behavior

If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. M97 (see “Machining small contour steps: M97” on page 193) can be used to prohibit the error message, but 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.



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) after 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 G41 or G42. M120 is then effective from this block until

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

M120 becomes effective at the start of block.

Limitations

- After an external or internal stop, you can only re-enter the contour with the function RESTORE POS. AT N.
- 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

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 blocks

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

```
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.

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

Standard behavior

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

Behavior with M104

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

Input

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

Effect

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

M140 becomes effective at the start of the block.

Example NC blocks

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

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

```
N45 G01 X+0 Y+38.5 F125 M140 MB 50
N55 G01 X+0 Y+38.5 F125 M140 MB MAX
```

M140 is also effective if the tilted-working-plane function, M114 or M128 is active. On machines with tilting heads, the TNC then moves the tool in the tilted coordinate system.

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

Suppressing touch probe monitoring: M141

Standard behavior

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

Behavior with M141

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

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

M141 functions only for movements with straight-line blocks.

Effect

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

M141 becomes effective at the start of the block.

Delete modal program information: M142

Standard behavior

The TNC resets modal program information in the following situations:

- Select a new program.
- Execute a miscellaneous function M02, M30, or an N999999 %... block (depending on MP7300).
- Defining cycles for basic behavior with a new value

Behavior with M142

All modal program information except for basic rotation, 3-D rotation and Q parameters are reset.

Effect

M142 is effective only in the block in which it is programmed.

M142 becomes effective at the start of the block.

Delete basic rotation: M143

Standard behavior

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

Behavior with M143

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

Effect

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

M143 becomes effective at the start of the block.

7.5 Miscellaneous Functions for Rotary Axes

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

Standard behavior

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

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

Feed rate in mm/min on rotary axes with M116

The machine geometry must be entered in Machine Parameters 7510 and following 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 block. With a rotary axis, the feed rate is not changed during execution of the block even if the tool moves toward the center of the rotary axis.

Effect

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

M116 becomes effective at the start of block.

Shorter-path traverse of rotary axes: M126

Standard behavior

The standard behavior of the TNC while positioning rotary axes whose display has been reduced to values less than 360° depends on Machine Parameter 7682. In MP7682 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. Examples:

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

Behavior with M126

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

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

Effect

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

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

Standard behavior

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

Example:

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

Behavior with M94

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

Example NC blocks

To reduce display of all active rotary axes:

```
N50 M94 *
```

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 it is programmed.

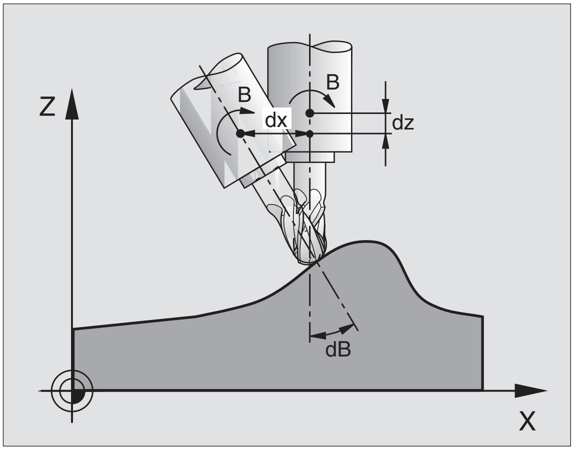
M94 becomes effective at the start of block.

Automatic compensation of machine geometry when working with tilted axes: M114

The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.

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 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 G41/G42 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 your 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.

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

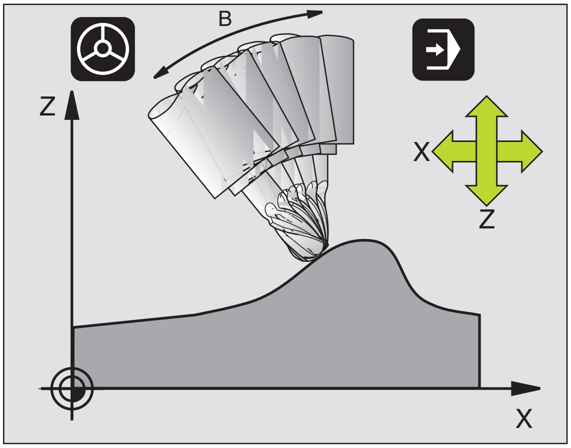
The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.

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 for 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 system is possible when M128 is active.

For tilted axes with Hirth coupling: Do not change the position of the tilted axis until 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 MP7471, the feed rate from MP7471 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.

If M128 is active, the TNC shows in the status display the following symbol: .

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° (through a positioning command or datum shift) 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.

M128 with 3-D tool compensation

If you carry out a 3-D tool compensation with active M128 and active radius compensation G41/G42, the TNC will automatically position the rotary axes for certain machine geometries (Peripheral milling, see "Peripheral Milling: 3-D radius compensation with workpiece orientation," page 146).

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.

Example NC blocks

Feed rate of 1000 mm/min for compensation movements.

```
G01 G41 X+0 Y+38.5 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 MP7440 you can change the standard behavior of the TNC so that M134 becomes active automatically whenever a program is selected (see "General User Parameters," page 466).

Behavior with M134

The TNC 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.

Selecting tilting axes: M138

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 blocks

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

```
G00 G40 Z+100 M138 C *
```

Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block: M144

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.

Behavior with M144

The TNC calculates into the position value any changes in the machine's kinematic configuration which result, for example, from adding a spindle attachment. If the position of a controlled tilted axis changes, the position of the tool tip to the workpiece is also changed. The resulting offset is calculated in the position display.

> Positioning blocks with M91/M92 are permitted if M144 is active.
>
> The position display in the operating modes FULL SEQUENCE and SINGLE BLOCK does not change until the tilting axes have reached their final position.

Effect

M144 becomes effective at the start of the block. M144 does not function in connection with M114, M128 or a tilted working plane.

You can cancel M144 by programming M145.

> The machine geometry must be entered in Machine Parameters 7502 and following by the machine tool builder. The machine tool builder determines the behavior in the automatic and manual operating modes. Refer to your machine manual.

7.6 Miscellaneous Functions for Laser Cutting Machines

Principle

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 the block.

Output the programmed voltage directly: M200

Behavior with M200

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

Input range: 0 to 9999 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

Behavior with 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 9999 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

Behavior with 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

Behavior with 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

Behavior with 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 Working with Cycles

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and other special cycles are also provided as standard cycles (see table on next page).

Fixed cycles with numbers 200 and above use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q200 is always assigned the set-up clearance, Q202 the plunging depth, etc.

> In order to avoid erroneous entries during cycle definition, you should run a graphical program test before machining (see “Test Run” on page 424).

Defining a cycle using soft keys

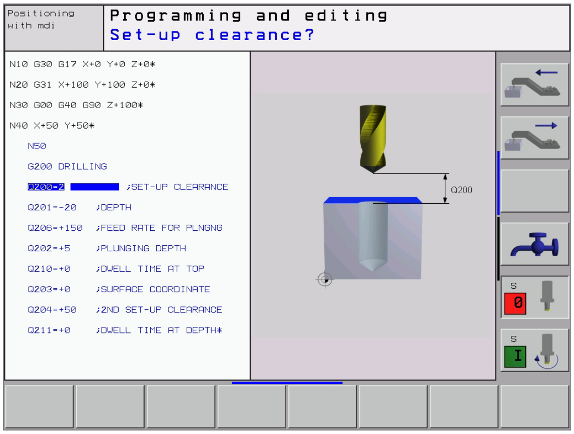
CYCL DEF

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

DRILLING/ THREAD

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

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



Example NC block

```
N10 G200 DRILLING
     Q200=2        ;SET-UP CLEARANCE
     Q201=3        ;DEPTH
     Q206=150      ;FEED RATE FOR PLUNGING
     Q202=5        ;PLUNGING DEPTH
     Q210=0        ;DWELL TIME AT TOP
     Q203=+0       ;SURFACE COORDINATE
     Q204=50       ;2ND SET-UP CLEARANCE
     Q211=0.25     ;DWELL TIME AT DEPTH
```

Group of cycles	Soft key
Cycles for pecking, reaming, boring, counterboring, tapping and thread milling	DRILLING/ THREAD
Cycles for Milling Pockets, Studs and Slots	POCKETS/ STUDS/ SLOTS
Cycles for producing 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	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, oriented spindle stop and tolerance	SPECIAL CYCLES

If you use indirect parameter assignments in fixed cycles with numbers greater than 200 (e.g. **D00 Q210 = Q1**), any change in the assigned parameter (e.g. Q1) will have no effect after the cycle definition. Define the cycle parameter (e.g. **D00 Q210**) directly in such cases.

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 setup clearance and plunging depth.

Calling a 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 detailed in the descriptions for each cycle.

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

- Cycle G220 for circular and Cycle G221 for linear hole patterns
- SL Cycle G14 CONTOUR GEOMETRY
- SL Cycle G20 CONTOUR DATA
- Cycle G62 TOLERANCE
- Coordinate transformation cycles
- Cycle G04 DWELL TIME

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

Calling a cycle with G79 (CYCL CALL)

The **G79** function calls the last defined fixed cycle once. The starting point of the cycle is the position that was programmed last before the G79 block.

CYCL CALL

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

Calling a cycle with G79 PAT (CYCL CALL PAT)

The **G79 PAT** function calls the most recently defined fixed cycle at all positions defined in a point table (see “Point Tables” on page 218).

Calling a cycle with G79:G01 (CYCL CALL POS)

The **G79:G01** function calls the fixed cycle that was last defined. The starting point of the cycle is the position that you defined in the **G79:G01** block.

> The TNC first moves the tool to the defined position and then calls the fixed cycle most recently defined.
>
> The feed rate most recently defined in the **G79:G01** block applies only for traverse to the start position programmed in this block.
>
> As a rule, the TNC moves without radius compensation (R0) to the position defined in the **G79:G01** block.
>
> If you use **G79:G01** to call a cycle in which a start position is defined (for example Cycle 212), then the TNC uses the position defined in the **G79:G01** block as starting position.

Cycle call with M99/89

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

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

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

- **M99** in the positioning block in which you move to the last starting point, or
- **G79**, or
- Define with **CYCL DEF** a new fixed 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 principal 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

Function

You should create a point table whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Creating a point table

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

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

FILE NAME ?

NEW.PNT ENT — Enter the name and file type of the point table 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 blocks window and displays an empty point table.

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

Repeat the process until all desired coordinates have been entered.

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

Selecting a point table in the program

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

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

POINT TABLE — Press the POINT TABLE soft key.

Enter the name of the point table and confirm your entry with the ENT key.

Example NC block

```
N72 %:PAT: "NAMES"*
```

Calling a cycle in connection with point tables

> With **G79 PAT** the TNC runs the point 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).
- If required, enter a miscellaneous function M, then confirm with the END key.

The TNC moves the tool back to the clearance height over each successive starting point (clearance height = the spindle axis coordinate for cycle call). To use this procedure for cycles above Cycle 199, you must define the 2nd set-up clearance (Q204) to equal 0.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103 (see “Feed rate factor for plunging movements: M103” on page 194).

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.

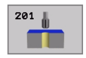
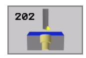
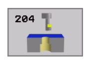
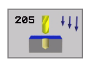
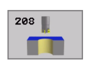
Effect of the point tables with Cycles 210 to 215

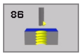
The TNC interprets the points as an additional datum shift. If you want to use the points defined in the point table as starting-point coordinates, you must define the starting points and the workpiece surface coordinate (Q203) in the respective milling cycle as 0.

8.3 Cycles for Drilling, Tapping and Thread Milling

Overview

The TNC offers 19 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 set-up clearance, chip breaking, and decrementing	
G204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	
G205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	
G208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance	

Cycle	Soft key
G84 TAPPING With a floating tap holder	
G85 RIGID TAPPING Without a floating tap holder	
G86 THREAD CUTTING For integration into OEM cycles	
G206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	
G207 RIGID TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	
G209 TAPPING W/ CHIP BRKG Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	
G262 THREAD MILLING Cycle for milling a thread in pre-drilled material	
G263 THREAD MLLNG/CNTSNKG Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	
G264 THREAD DRILLING/MLLNG Cycle for drilling into the solid material with subsequent milling of the thread with a tool	
G265 HEL.THREAD DRLG/MLG Cycle for milling the thread into the solid material	
G267 OUTSIDE THREAD MLLNG Cycle for milling an external thread and machining a countersunk chamfer	

PECKING (Cycle G83)

1 The tool drills from the current position to the first plunging depth at the programmed feed rate F.

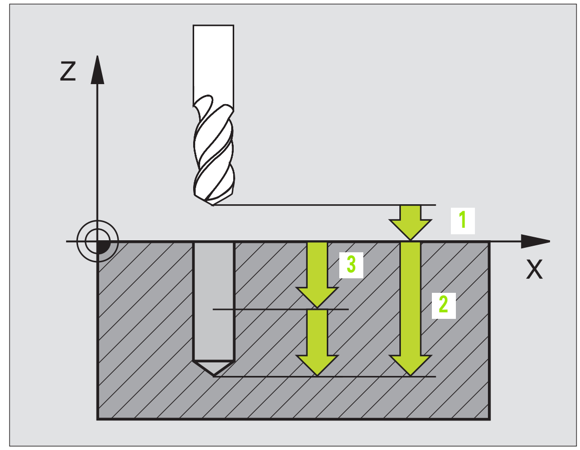
2 When it reaches the first plunging depth, the tool retracts at rapid traverse to the starting position and advances again to the first plunging depth minus the advanced stop distance t.

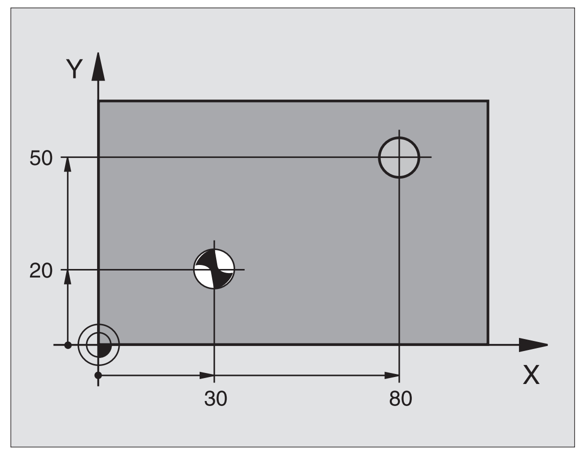
3 The advanced stop distance is automatically calculated by the control:
- At a total hole depth of up to 30 mm: *t* = 0.6 mm
- At a total hole depth exceeding 30 mm: *t* = hole depth / 50
- Maximum advanced stop distance: 7 mm

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 depth is reached.

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



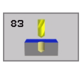


Before programming, note the following:

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



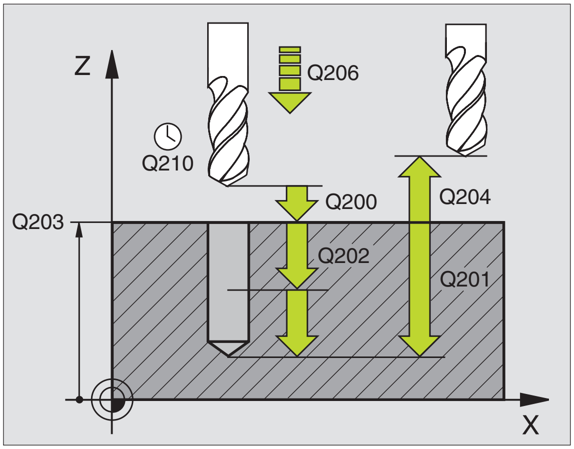
- **Set-up clearance** 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **Total hole depth** 2 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **Plunging depth** 3 (incremental value): Infeed per cut The total hole depth does not have to be a multiple of the plunging depth. The tool will drill to the total hole depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the total hole depth
- **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.

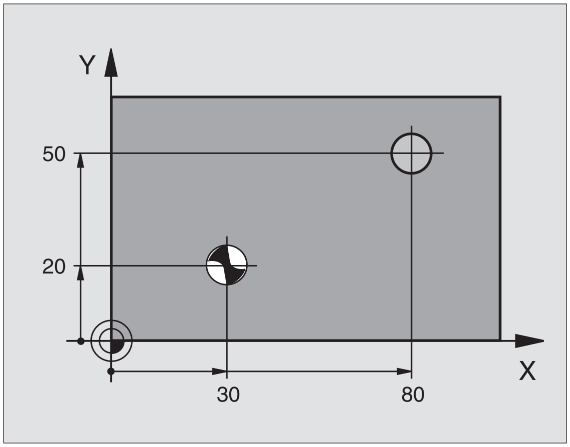
Example: NC block

```
N10 G83 P01 2  P02 -20  P03 -8  P04 0
P05 500*
```

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 depth is reached.
6 The tool is retracted from the hole bottom to the set-up clearance or, if programmed, to the 2nd set-up clearance at rapid traverse. 2nd set-up clearance





Before programming, note the following:

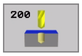
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



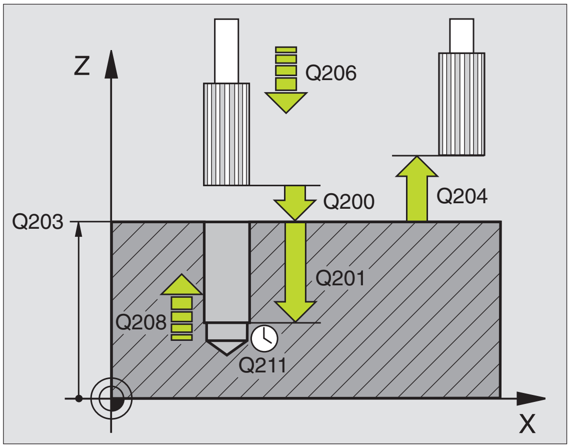
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- **Feed rate for plunging** Q206: Traversing speed of the tool during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

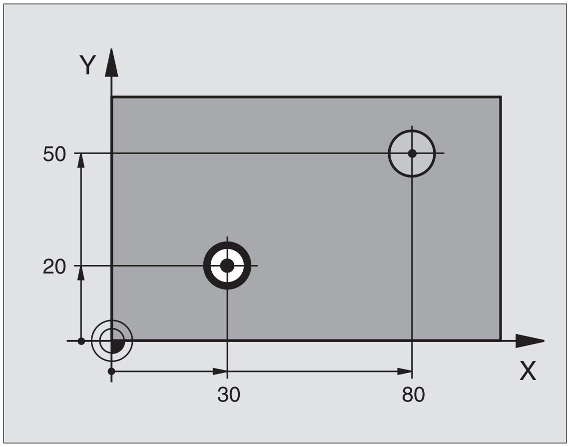
Example: NC blocks

```
N100 G00 Z+100 G40
N110 G200 DRILLING
     Q200=2        ;SET-UP CLEARANCE
     Q291=-15      ;DEPTH
     Q206=250      ;FEED RATE FOR PLUNGING
     Q202=5        ;PLUNGING DEPTH
     Q210=0        ;DWELL TIME AT TOP
     Q203=+20      ;SURFACE COORDINATE
     Q204=100      ;2ND SET-UP CLEARANCE
     Q211=0.1      ;DWELL TIME AT DEPTH
N120 X+30 Y+20 M3 M99
N130 X+80 Y+50 M99
N140 Z+100 M2
```

REAMING (Cycle G201)

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





Before programming, note the following:

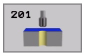
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



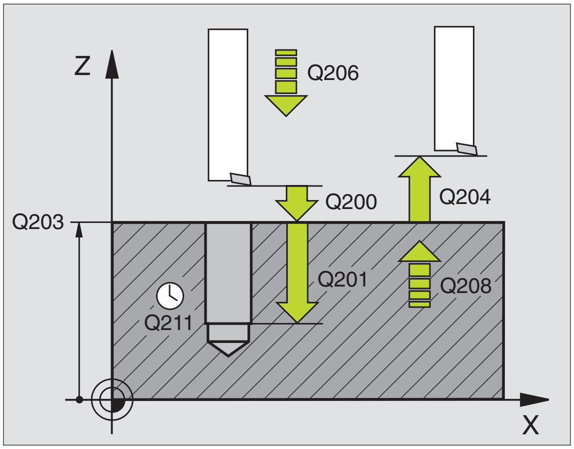
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for plunging** Q206: Traversing speed of the tool during reaming in mm/min.
- **Dwell time at depth**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

Example: NC blocks

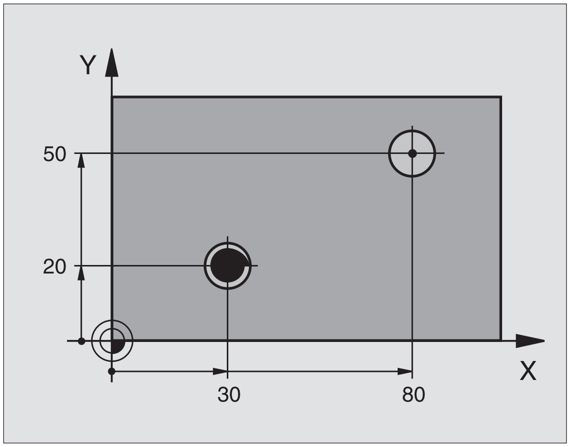
```
N100 G00 Z+100 G40
N110 G201 REAMING
     Q200=2          ;SET-UP CLEARANCE
     Q201=-15        ;DEPTH
     Q206=100        ;FEED RATE FOR PLUNGING
     Q211=0.5        ;DWELL TIME AT DEPTH
     Q208=250        ;RETRACTION FEED RATE
     Q203=+20        ;SURFACE COORDINATE
     Q204=100        ;2ND SET-UP CLEARANCE
N120 X+30 Y+20 M3 M99
N130 X+80 Y+50 M99
N140 G00 Z+100 M2
```

BORING (Cycle G202)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle G202.



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 programmed depth at the feed rate for plunging.
3. If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4. The TNC then orients the spindle to the position that is defined in parameter **Q336.**
5. If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
6. The TNC moves the tool at the retraction feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at 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.**

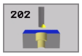
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for plunging** Q206: Traversing speed of the tool during boring in mm/min.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at feed rate for plunging.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Disengaging direction** (0/1/2/3/4) Q214: Determine the direction in which the TNC retracts the tool at the hole bottom (after spindle orientation).

0: Do not retract tool
1: Retract tool in the negative reference axis direction
2: Retract tool in the negative secondary axis direction
3: Retract tool in the positive reference axis direction
4: Retract tool in the positive secondary axis direction

Danger of collision

Select a disengaging direction in which the tool moves away from the edge of the hole.

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

During retraction the TNC automatically takes an active rotation of the coordinate system into account.

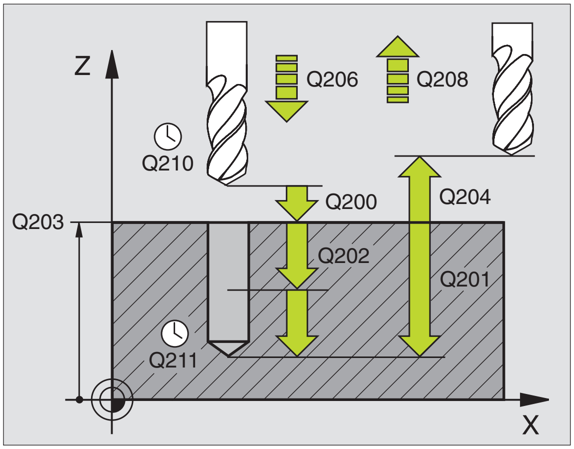
- **Angle for spindle orientation** Q336 (absolute value): Angle at which the TNC positions the tool before retracting it.

Example:

```
N100 G00 Z+100 G40
N110 G202 BORING
     Q200=2        ;SET-UP CLEARANCE
     Q201=-15      ;DEPTH
     Q206=100      ;FEED RATE FOR PLUNGING
     Q211=0.5      ;DWELL TIME AT DEPTH
     Q208=250      ;RETRACTION FEED RATE
     Q203=+20      ;SURFACE COORDINATE
     Q204=100      ;2ND SET-UP CLEARANCE
     Q214=1        ;DISENGAGING DIRECTN
     Q336=0        ;ANGLE OF SPINDLE
N120 X+30 Y+20 M3
N130 G79
N140 L X+80 Y+50 FMAX M99
```

UNIVERSAL DRILLING (Cycle G203)

1. The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.
2. The tool drills to the first plunging depth at the programmed feed rate F.
3. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool retracts at the retraction feed rate to set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at rapid traverse to the set-up clearance above the first PLUNGING DEPTH.
4. The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
5. The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6. The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to the set-up clearance at the retraction feed rate. If 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 DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min.

Example: NC blocks

```
N110 G203 UNIVERSAL DRILLING
    Q200=2        ;SET-UP CLEARANCE
    Q201=-20      ;DEPTH
    Q206=150      ;FEED RATE FOR PLUNGING
    Q202=5        ;PLUNGING DEPTH
    Q210=0        ;DWELL TIME AT TOP
    Q203=+20      ;SURFACE COORDINATE
    Q204=50       ;2ND SET-UP CLEARANCE
    Q212=0.2      ;DECREMENT
    Q213=3        ;BREAKS
    Q205=3        ;MIN. PLUNGING DEPTH
    Q211=0.25     ;DWELL TIME AT DEPTH
    Q208=500      ;RETRACTION FEED RATE
    Q256=0.2      ;DIST. FOR CHIP BRKNG
```

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

BACK BORING (Cycle G204)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Special boring bars for upward cutting are required for this cycle.

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

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

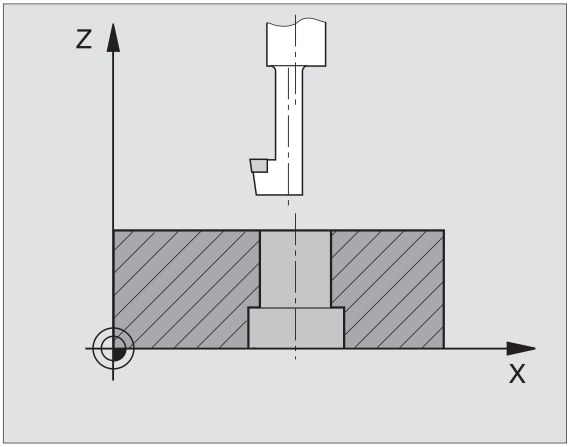
Before programming, note the following:

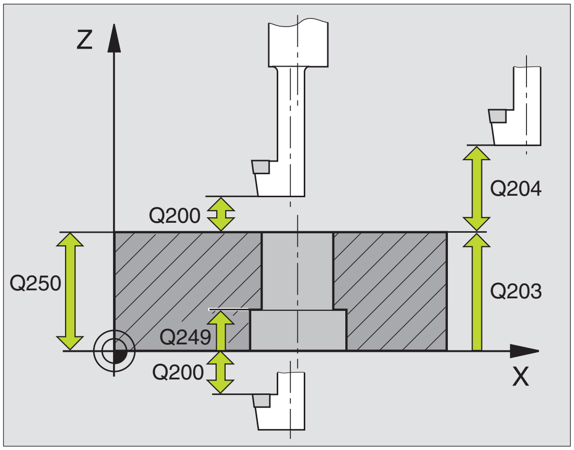
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

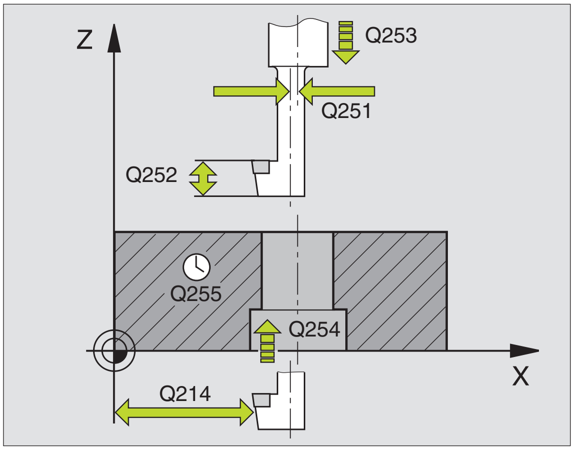
The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

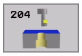
The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

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 reference axis direction
2: Displace tool in the negative secondary axis direction
3: Displace tool in the positive reference 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 the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. Select a disengaging direction in which the tool moves away from the edge of the hole.

- **Angle for spindle orientation** Q336 (absolute value): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole.

Example: NC blocks

```
N110 G204 BACK BORING
     Q200=2          ;SET-UP CLEARANCE
     Q249=+5         ;DEPTH OF COUNTERBORE
     Q250=20         ;MATERIAL THICKNESS
     Q251=3.5        ;OFF-CENTER DISTANCE
     Q252=15         ;TOOL EDGE HEIGHT
     Q253=750        ;F PRE-POSITIONING
     Q254=200        ;F COUNTERBORING
     Q255=0          ;DWELL TIME
     Q203=+20        ;SURFACE COORDINATE
     Q204=50         ;2ND SET-UP CLEARANCE
     Q214=1          ;DISENGAGING DIRECTN
     Q336=0          ;ANGLE OF SPINDLE
```

UNIVERSAL PECKING (Cycle G205)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate F.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to set-up 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 the 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 DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

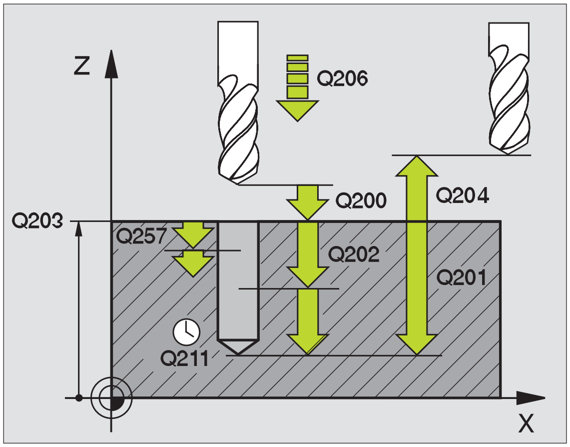
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

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

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

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



Example: NC blocks

```
N110 G205 UNIVERSAL PECKING
     Q200=2          ;SET-UP CLEARANCE
     Q201=-80        ;DEPTH
     Q206=150        ;FEED RATE FOR PLUNGING
     Q202=15         ;PLUNGING DEPTH
     Q203=+100       ;SURFACE COORDINATE
     Q204=50         ;2ND SET-UP CLEARANCE
     Q212=0.5        ;DECREMENT
     Q205=3          ;MIN. PLUNGING DEPTH
     Q258=0.5        ;UPPER ADV STOP DISTANCE
     Q259=1          ;LOWER ADV STOP DIST
     Q257=5          ;DEPTH FOR CHIP BRKNG
     Q256=0.2        ;DIST. FOR CHIP BRKNG
     Q211=0.25       ;DWELL TIME AT DEPTH
```

BORE MILLING (Cycle G208)

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 mills in a helix from the current position to the first plunging depth at the programmed feed rate.
3 When the drilling depth is reached, the TNC once again traverses a full circle to remove the material remaining after the initial plunge.
4 The TNC then positions the tool at the center of the hole again.
5 Finally the TNC returns to the set-up clearance at 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.**

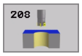
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

If you have entered the bore hole diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

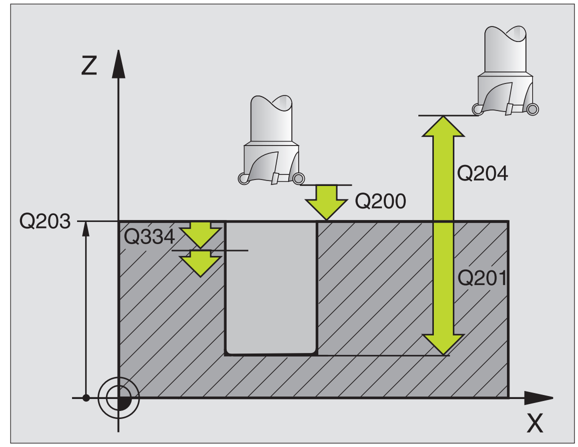
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

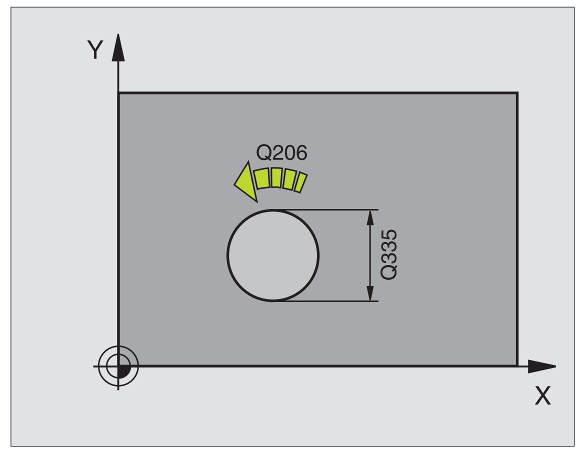


- **`Set-up clearance`** Q200 (incremental value): Distance between tool lower edge and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool during helical drilling in mm/min.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

> Note that if the infeed distance is too large, the tool or the workpiece may be damaged.
>
> To prevent the infeeds from being too large, enter the maximum plunge angle of the tool in the **`ANGLE`** column of the tool table (see "Tool Data," page 131). The TNC then automatically calculates the max. infeed permitted and changes your entered value accordingly.

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



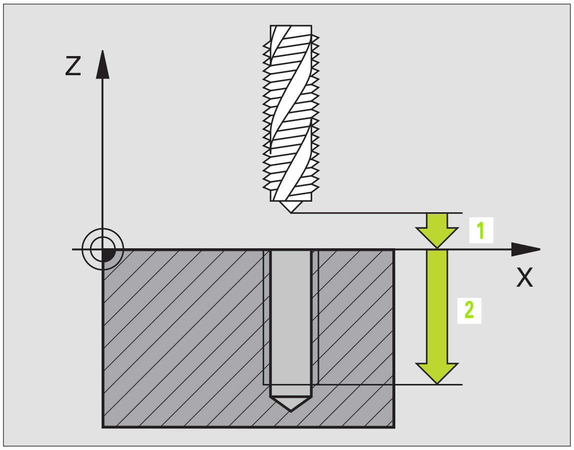


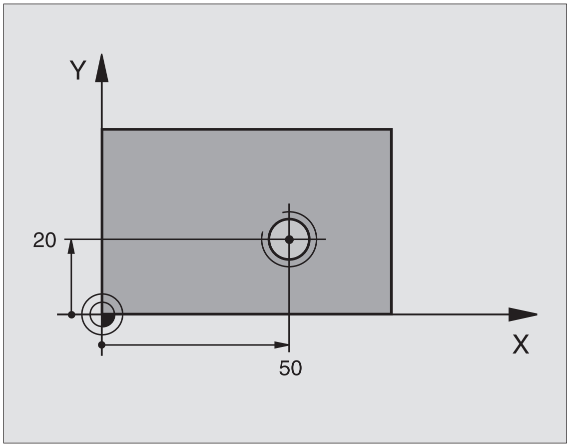
Example: NC blocks

```
N120 G208 BORE MILLING
     Q200=2          ;SET-UP CLEARANCE
     Q201=-80        ;DEPTH
     Q206=150        ;FEED RATE FOR PLUNGING
     Q334=1.5        ;PLUNGING DEPTH
     Q203=+100       ;SURFACE COORDINATE
     Q204=50         ;2ND SET-UP CLEARANCE
     Q335=25         ;NOMINAL DIAMETER
     Q342=0          ;ROUGHING DIAMETER
```

TAPPING 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`**.

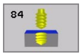
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

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



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

Example: NC block

```
N13 G84 P01 2  P02 -20  P03 0  P04 100 *
```

The feed rate is calculated as follows: F = S x p

F Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.

TAPPING NEW with floating tap holder (Cycle G206)

1. The TNC positions the tool in the tool axis at rapid traverse 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 rapid traverse.
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 cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

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

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

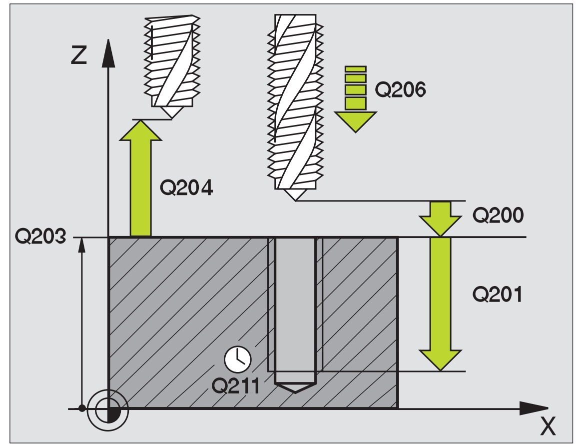
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch.
- **`Total hole depth`** Q201 (thread length, incremental value): Distance between workpiece surface and end of thread.
- **`Feed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

The feed rate is calculated as follows: F = S x p

F Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.



Example: NC blocks

```
N250 G206 TAPPING NEW
    Q200=2        ;SET-UP CLEARANCE
    Q201=-20      ;DEPTH
    Q206=150      ;FEED RATE FOR PLUNGING
    Q211=0.25     ;DWELL TIME AT DEPTH
    Q203=+25      ;SURFACE COORDINATE
    Q204=50       ;2ND SET-UP CLEARANCE
```

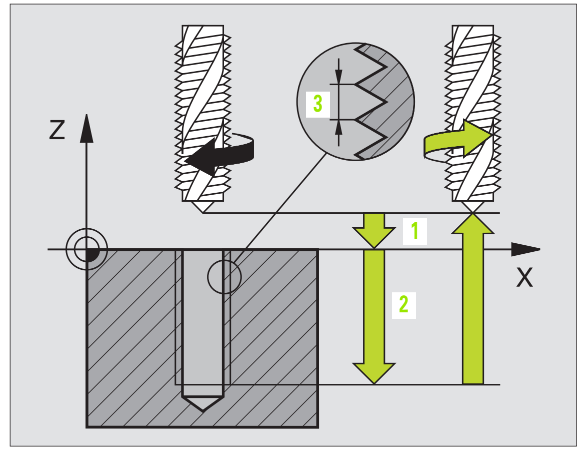
RIGID TAPPING (Cycle G85)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

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

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 MP7160).
- 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 total hole depth parameter determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

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

Example: NC block

```
N18 G85 P01  2 P02 -20 P03 +1 *
```

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

Retracting after a program interruption

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.

RIGID TAPPING NEW (Cycle G207)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

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

See "RIGID TAPPING (Cycle G85)," page 242, for the advantages that rigid tapping offers over tapping with a floating tap holder.

1 The TNC positions the tool in the tool axis at rapid traverse 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 rapid traverse.
4 The TNC stops the spindle rotation at the 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 total hole depth parameter determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

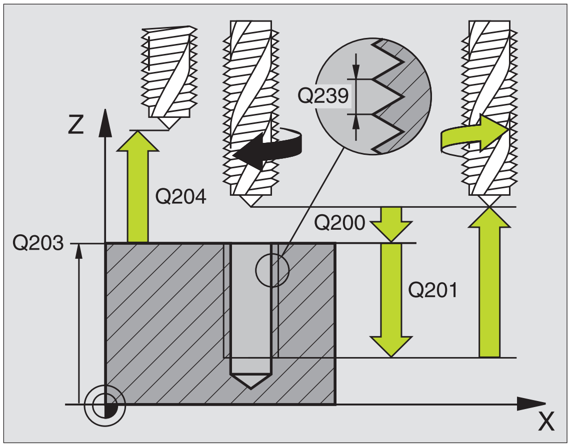
At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

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



Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. 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          ;SET-UP CLEARANCE
    Q201=-20        ;DEPTH
    Q239=+1         ;PITCH
    Q203=+25        ;SURFACE COORDINATE
    Q204=50         ;2ND SET-UP CLEARANCE
```

THREAD CUTTING (Cycle G86)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

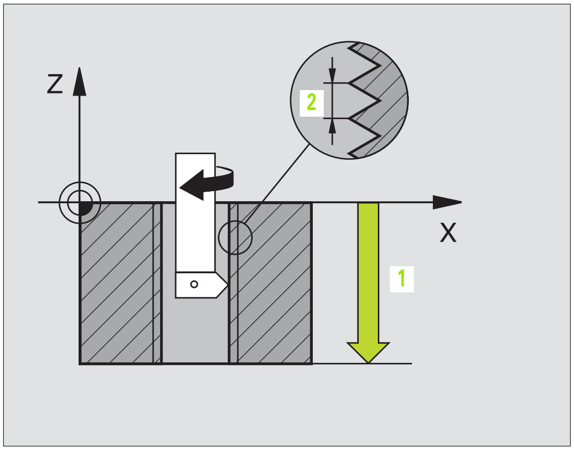
Cycle G86 THREAD CUTTING is performed by means of spindle control. The tool moves with the active spindle speed from its current position to the entered depth. As soon as it reaches the end of thread, spindle rotation is stopped. Tool approach and departure must be programmed separately. The most convenient 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.



Example: NC block

```
N22 G86 P01 -20 P02 +1 *
```

- **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)

TAPPING WITH CHIP BREAKING (Cycle G209)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

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

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

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

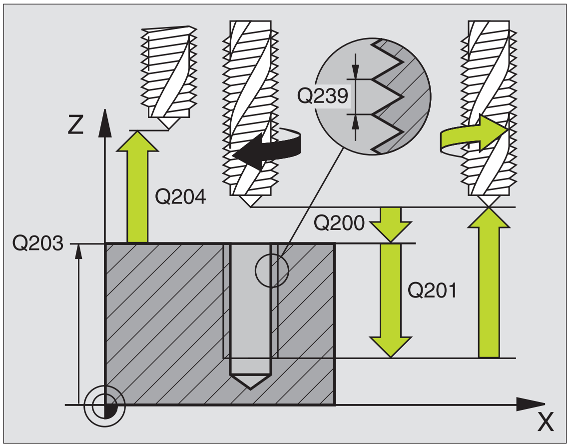
At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

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



Example: NC blocks

```
N260 G207 RIGID TAPPING NEW
    Q200=2         ;SET-UP CLEARANCE
    Q201=-20       ;DEPTH
    Q239=+1        ;PITCH
    Q203=+25       ;SURFACE COORDINATE
    Q204=50        ;2ND SET-UP CLEARANCE
```

Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. 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.

Fundamentals of thread milling

Prerequisites

- Your machine tool should feature internal spindle cooling (cooling lubricant at least 30 bar, compressed air supply at least 6 bar).
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius DR in the tool call .
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265 you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread / – = left-hand thread) and milling method Q351 (+1 = climb / –1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

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

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

Danger of collision!

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

Procedure in case of a tool break

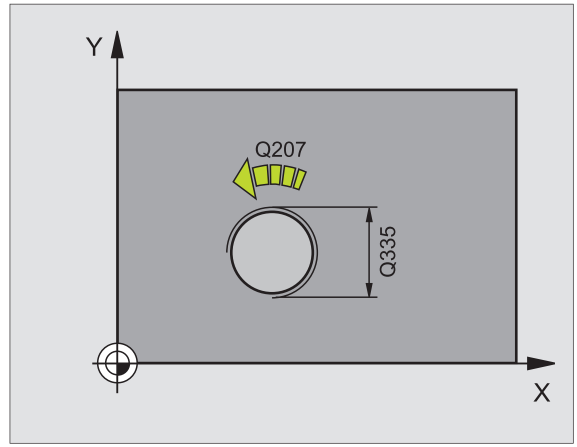
If a tool break occurs during thread cutting, stop the program run, change to the Positioning with MDI operating mode and move the tool in a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.

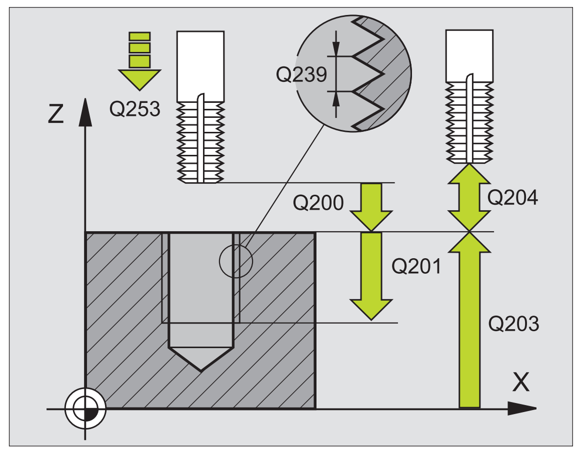
The TNC references the programmed feed rate during thread milling to the tool cutting edge. Since the TNC, however, always displays the feed rate relative to the path of the tool tip, the displayed value does not match the programmed value.

The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRRORING in only one axis.

THREAD MILLING (Cycle G262)

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



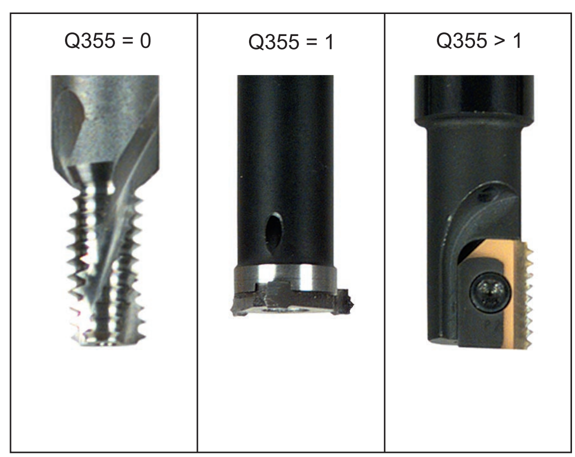


Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter thread depth determines the working direction. If you program the thread DEPTH = 0, the cycle will not be executed.

The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the pitch of the tool diameter is four times smaller than the nominal thread diameter.



Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **Nominal diameter** Q335: Nominal thread diameter.
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **Threads per step** Q355: Number of thread revolutions by which the tool is offset (see figure at lower right):
  0 = one 360° helical line to the thread depth
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **Climb or up-cut** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G262 THREAD MILLING
     Q335=10         ;NOMINAL DIAMETER
     Q239=+1.5       ;PITCH
     Q201=-20        ;THREAD DEPTH
     Q355=0          ;THREADS PER STEP
     Q253=750        ;F PRE-POSITIONING
     Q351=+1         ;CLIMB OR UP-CUT
     Q200=2          ;SET-UP CLEARANCE
     Q203=+30        ;SURFACE COORDINATE
     Q204=50         ;2ND SET-UP CLEARANCE
     Q207=500        ;FEED RATE FOR MILLING
```

THREAD MILLING/COUNTERSINKING (Cycle G263)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.

Countersinking

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

Countersinking at front

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

Thread milling

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

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

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

11 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Countersinking depth
3rd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

If you wish to countersink with the front of the tool, define the countersinking depth as 0.

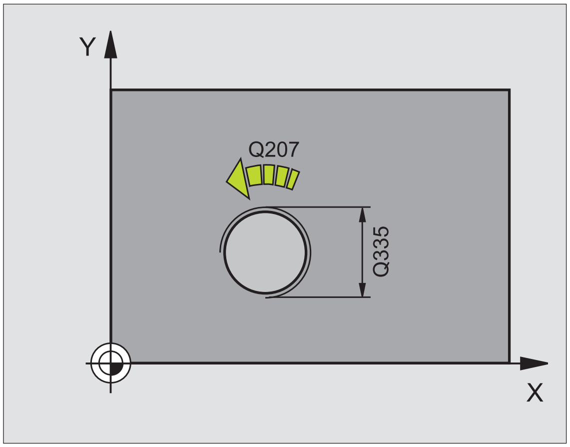
Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

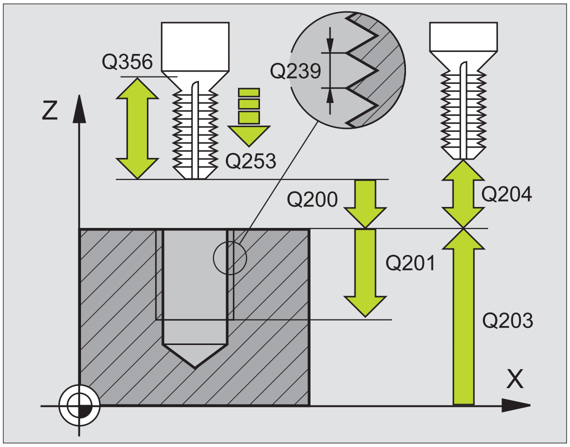
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

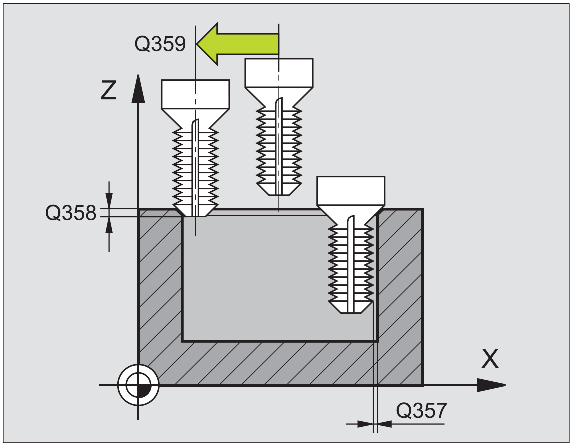
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **Nominal diameter** Q335: Nominal thread diameter.
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **Countersinking depth** Q356 (incremental value): Distance between tool point and the top surface of the workpiece.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **Climb or up-cut** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Set-up clearance to the side** Q357 (incremental value): Distance between tool tooth and the wall.
- **Depth at front** Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- **Countersinking offset at front** Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.







- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G263 THREAD MLLNG/CNTSNKG
    Q335=10       ;NOMINAL DIAMETER
    Q239=+1.5     ;PITCH
    Q201=-16      ;THREAD DEPTH
    Q356=-20      ;COUNTERSINKING DEPTH
    Q253=750      ;F PRE-POSITIONING
    Q351=+1       ;CLIMB OR UP-CUT
    Q200=2        ;SET-UP CLEARANCE
    Q357=0.2      ;CLEARANCE TO SIDE
    Q358=+0       ;DEPTH AT FRONT
    Q359=+0       ;OFFSET AT FRONT
    Q203=+30      ;SURFACE COORDINATE
    Q204=50       ;2ND SET-UP CLEARANCE
    Q254=150      ;F COUNTERBORING
    Q207=500      ;FEED RATE FOR MILLING
```

THREAD DRILLING/MILLING (Cycle G264)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.

Drilling

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

3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to set-up 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.

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

Countersinking at front

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

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

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

Thread milling

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

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

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

12 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Total hole depth
3rd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

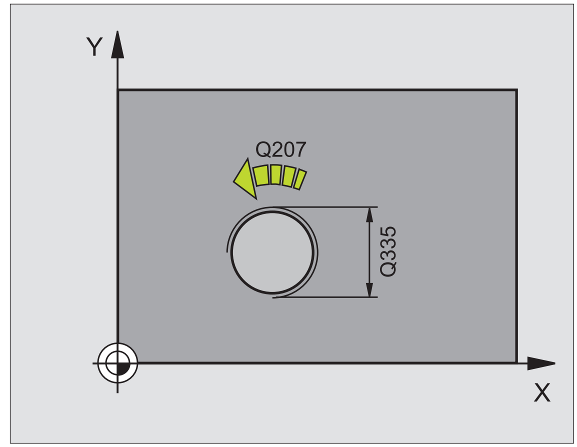
Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

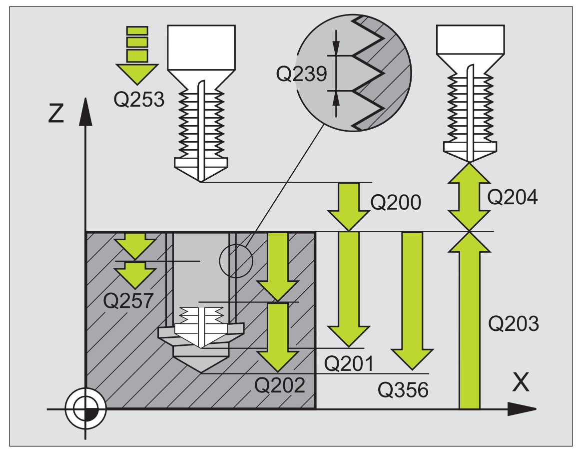
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

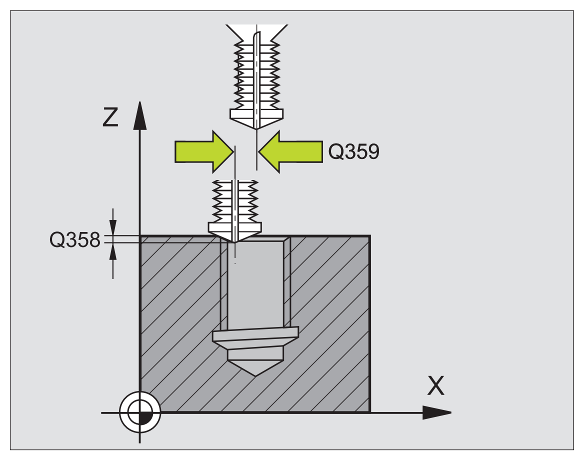
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **Nominal diameter** Q335: Nominal thread diameter.
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **Total hole depth** Q356 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **Climb or up-cut** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling
- **Plunging depth** Q202 (incremental value): Infeed per cut. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Upper advanced stop distance** Q258 (incremental value): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole.
- **Infeed depth for chip breaking** Q257 (incremental value): Depth at which TNC carries out chip breaking. There is no chip breaking if 0 is entered.
- **Retraction rate for chip breaking** Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.
- **Depth at front** Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- **Countersinking offset at front** Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.







- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool during drilling in mm/min.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G264 THREAD DRILLING/MILLING
     Q335=10        ;NOMINAL DIAMETER
     Q239=+1.5      ;PITCH
     Q201=-16       ;THREAD DEPTH
     Q356=-20       ;TOTAL HOLE DEPTH
     Q253=750       ;F PRE-POSITIONING
     Q351=+1        ;CLIMB OR UP-CUT
     Q202=5         ;PLUNGING DEPTH
     Q258=0.2       ;ADVANCED STOP DISTANCE
     Q257=5         ;DEPTH FOR CHIP BRKNG
     Q256=0.2       ;DIST. FOR CHIP BRKNG
     Q358=+0        ;DEPTH AT FRONT
     Q359=+0        ;OFFSET AT FRONT
     Q200=2         ;SET-UP CLEARANCE
     Q203=+30       ;SURFACE COORDINATE
     Q204=50        ;2ND SET-UP CLEARANCE
     Q206=150       ;FEED RATE FOR PLUNGING
     Q207=500       ;FEED RATE FOR MILLING
```

HELICAL THREAD DRILLING/MILLING (Cycle G265)

1. The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.

Countersinking at front

2. If countersinking is before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking is after thread milling, the tool moves at the feed rate for pre-positioning to the countersinking depth.
3. The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
4. The tool then moves on a semicircle to the hole center.

Thread milling

5. The tool moves at the programmed feed rate for pre-positioning to the starting plane for the thread.
6. The tool then approaches the thread diameter tangentially in a helical movement.
7. The tool moves on a continuous helical downward path until it reaches the thread depth.
8. After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

9 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

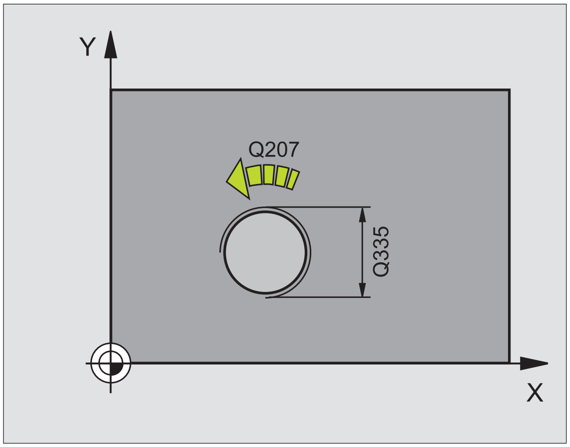
The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

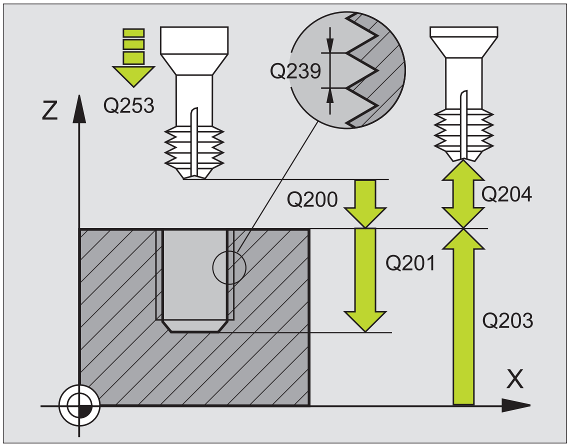
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

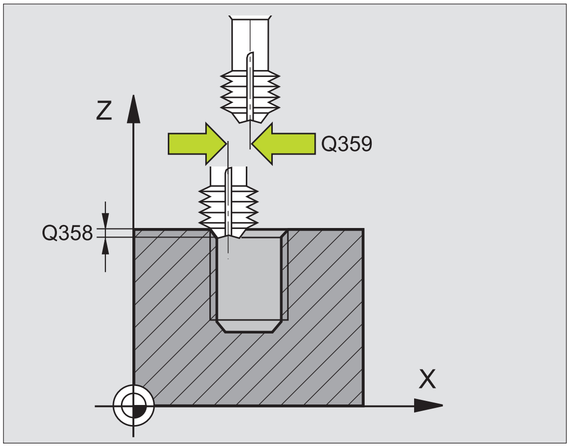
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **`Nominal diameter`** Q335: Nominal thread diameter.
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **`Thread depth`** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **`Depth at front`** Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- **`Countersinking offset at front`** Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.
- **`Countersink`** Q360: Execution of the chamfer
  0 = before thread machining
  1 = after thread machining
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.







- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Feed rate for counterboring`** Q254: Traversing speed of the tool during counterboring in mm/min.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G265 HEL. THREAD DRLG/MLG
    Q335=10        ;NOMINAL DIAMETER
    Q239=+1.5      ;PITCH
    Q201=-16       ;THREAD DEPTH
    Q253=750       ;F PRE-POSITIONING
    Q358=+0        ;DEPTH AT FRONT
    Q359=+0        ;OFFSET AT FRONT
    Q360=0         ;COUNTERSINKING
    Q200=2         ;SET-UP CLEARANCE
    Q203=+30       ;SURFACE COORDINATE
    Q204=50        ;2ND SET-UP CLEARANCE
    Q254=150       ;F COUNTERBORING
    Q207=500       ;FEED RATE FOR MILLING
```

OUTSIDE THREAD MILLING (Cycle G267)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.

Countersinking at front

2 The TNC moves in the reference axis of the working plane from the center of the stud to the starting point for countersinking at front. The position of the starting point is determined by the thread radius, tool radius and pitch.

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

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

5 The tool then moves on a semicircle to the starting point.

Thread milling

6 The TNC positions the tool to the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front.

7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.

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

9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset movements or in one continuous movement.

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

11 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (stud center) in the working plane with radius compensation **G40.**

The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

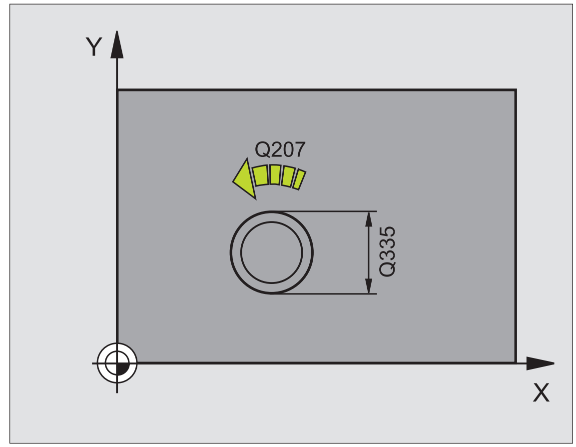
The algebraic sign for the cycle parameter thread depth determines the working direction.

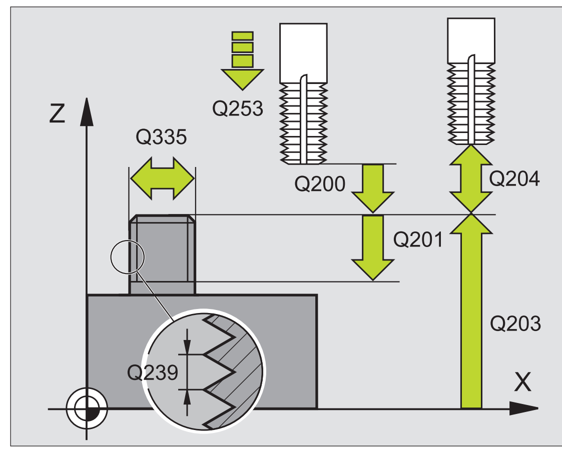
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

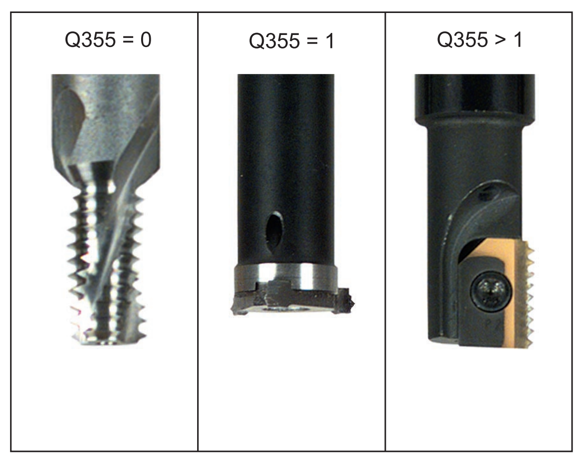
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **`Nominal diameter`** Q335: Nominal thread diameter.
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **`Thread depth`** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **`Threads per step`** Q355: Number of thread revolutions by which the tool is offset (see figure at lower right):
  0 = one helical line to the thread depth
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch.
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **`Climb or up-cut`** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling





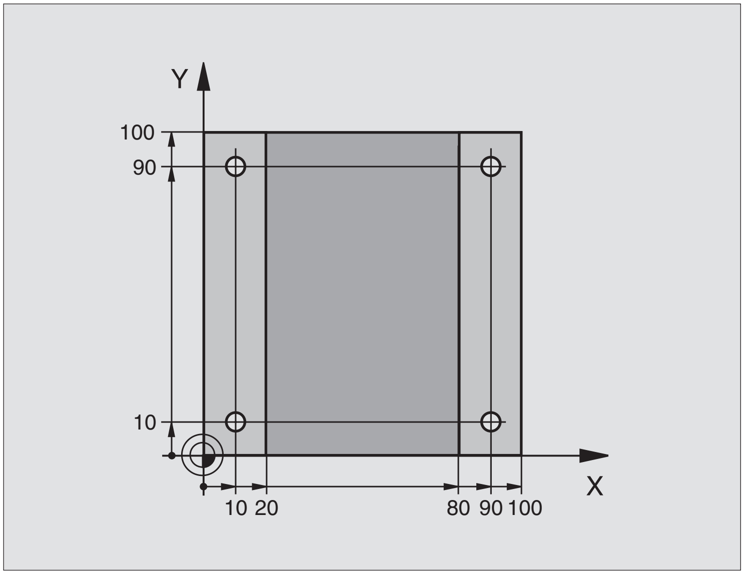


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth at front** Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- **Countersinking offset at front** Q359 (incremental value): Distance by which the TNC moves the tool center away from the stud center.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G267 OUTSIDE THREAD MLLNG
     Q335=10       ;NOMINAL DIAMETER
     Q239=+1.5     ;PITCH
     Q201=-20      ;THREAD DEPTH
     Q355=0        ;THREADS PER STEP
     Q253=750      ;F PRE-POSITIONING
     Q351=+1       ;CLIMB OR UP-CUT
     Q200=2        ;SET-UP CLEARANCE
     Q358=+0       ;DEPTH AT FRONT
     Q359=+0       ;OFFSET AT FRONT
     Q203=+30      ;SURFACE COORDINATE
     Q204=50       ;2ND SET-UP CLEARANCE
     Q254=150      ;F COUNTERBORING
     Q207=500      ;FEED RATE FOR MILLING
```

Example: Drilling cycles



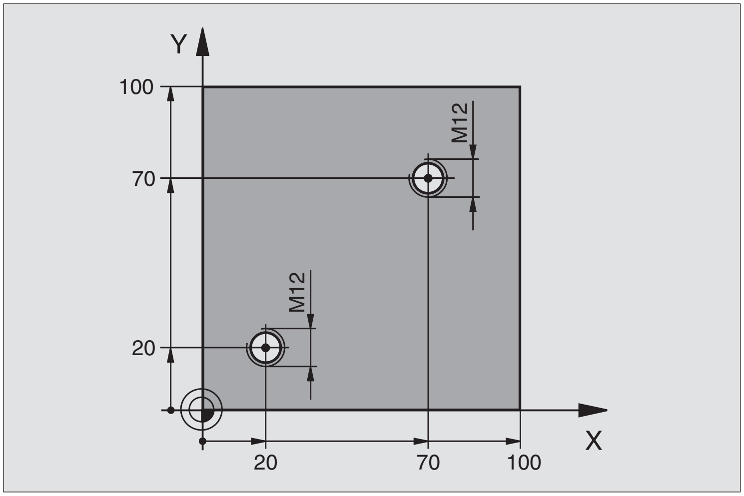
%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+3 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 DRILLING	Define cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	

`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 *`	Call the cycle

Example: Drilling cycles

Program sequence

- Program the drilling cycle in the main program
- Program machining within a subprogram (see "Subprograms," page 373)



`%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 *`	Define angle of spindle orientation
`N140 M19 *`	Orient spindle (makes it possible to cut repeatedly)
`N150 G01 G91 X-2 F1000 *`	Tool offset to prevent collision during tool infeed (depends
	on core diameter and tool)
`N160 G90 Z-30 *`	Move to starting depth
`N170 G91 X+2 *`	Reset the tool to hole center
`N180 G79 *`	Call Cycle 18
`N190 G90 Z+5 *`	Retract tool
`N200 G98 L0 *`	End of subprogram 1
`N999999 %C18 G71 *`	

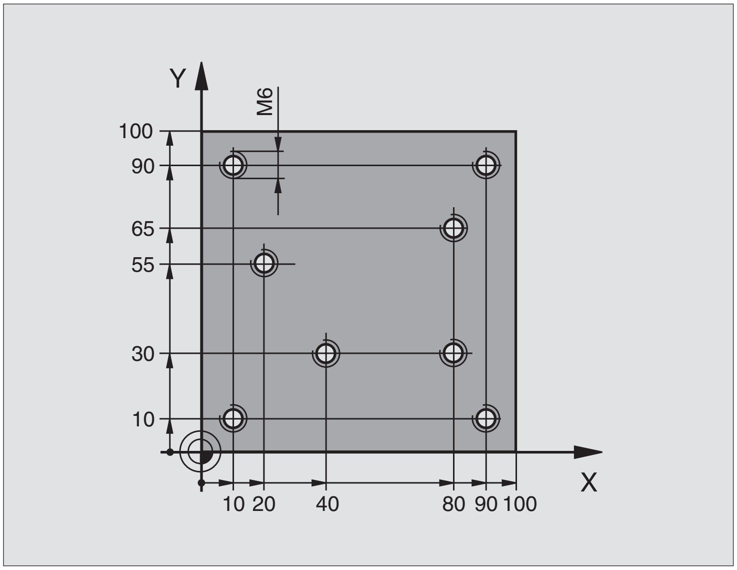
Example: Calling drilling cycles in connection with point tables

The drill hole coordinates are stored in the point table TAB1.PNT 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.

Program sequence

- Centering
- Drilling
- Tapping



%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 *	Define tool: 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 DRILLING	Cycle definition: Centering
Q200=2 ;SET-UP CLEARANCE	
Q201=-2 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=2 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	

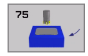
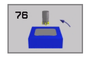
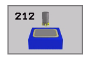
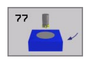
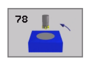
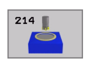
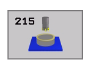
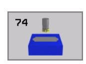
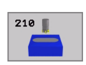
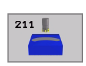
N100 G79 "PAT" F5000 M3 *	Cycle call in connection with point table TAB1.PNT
	Feed rate between points: 5000 mm/min
N110 G00 G40 Z+100 M6 *	Retract the tool, change the tool
N120 T2 G17 S5000 *	Call toll: drill
N130 G01 G40 Z+10 F5000 *	Move tool to clearance height (enter a value for F)
N140 G200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	
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 table TAB1.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]
```

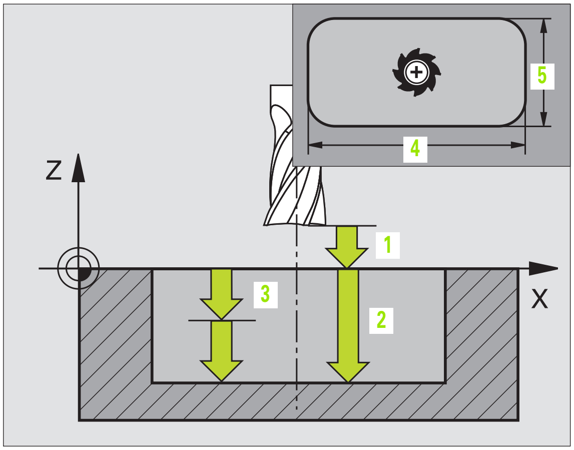
8.4 Cycles for Milling Pockets, Studs and Slots

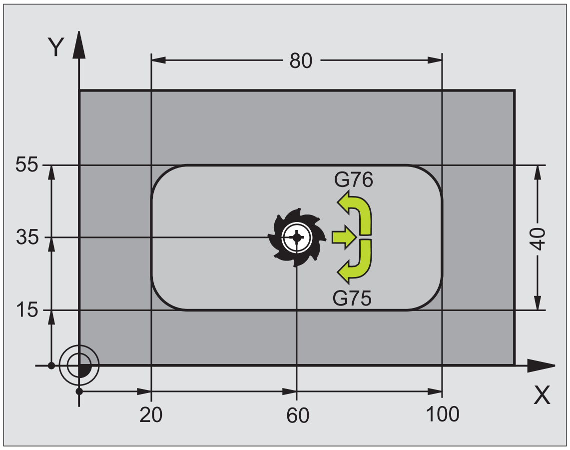
Overview

Cycle	Soft key
G75/G76 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning G75: In clockwise direction G76: In counterclockwise direction	 
G212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G77/G78 CIRCULAR POCKET MILLING Roughing cycle without automatic pre-positioning G77: In clockwise direction G78: In counterclockwise direction	 
G214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G74 SLOT MILLING Roughing/finishing cycle without automatic pre-positioning, vertical depth infeed	
G210 SLOT RECIP. PLNG Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed	
G211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed	

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:

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

Pre-position over the pocket center with radius compensation **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 DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

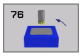
The following prerequisite applies for the 2nd side length: 2nd side length greater than [(2 x rounding radius) + stepover factor k].

Example: NC blocks

```
N27 G75 P01 2  P02 -20  P03 5  P04 100
    P05 X+80  P06 Y+40  P07 275 P08 5 *
...
N35 G76 P01 2  P02 -20  P03 5  P04 100
    P05 X+80  P06 Y+40  P07 275 P08 5 *
```

Direction of rotation during rough-out

- In clockwise direction: G75 (DR-)
- In counterclockwise direction: G76 (DR+)

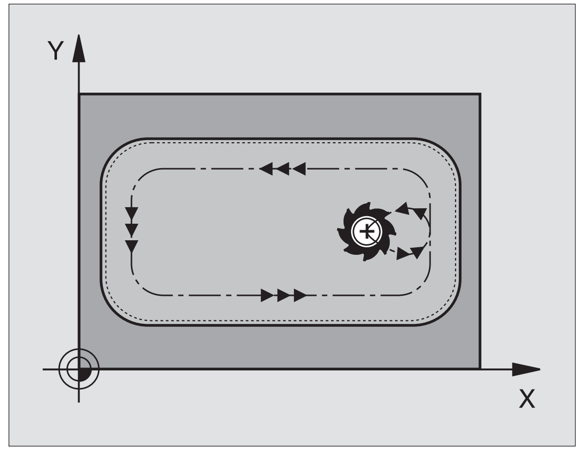


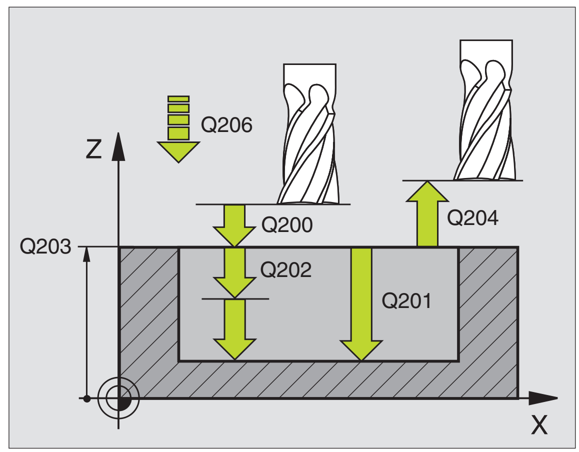
- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **Milling depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Feed rate for plunging:** Traversing speed of the tool during penetration
- **First side length 4** (incremental value): Pocket length, parallel to the reference axis of the working plane
- **2nd side length 5**: Pocket width
- **Feed rate F:** Traversing speed of the tool in the working plane.
- **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.

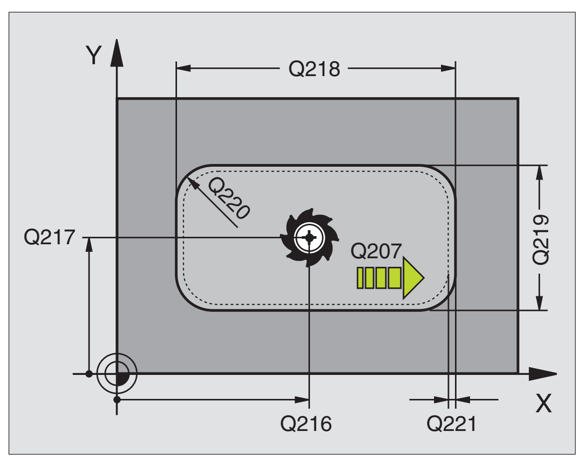
Calculations:
Stepover factor k = K x R

K: is the overlap factor, preset in MP7430, and
R: is the cutter radius

POCKET FINISHING (Cycle G212)







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

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

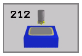
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

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

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



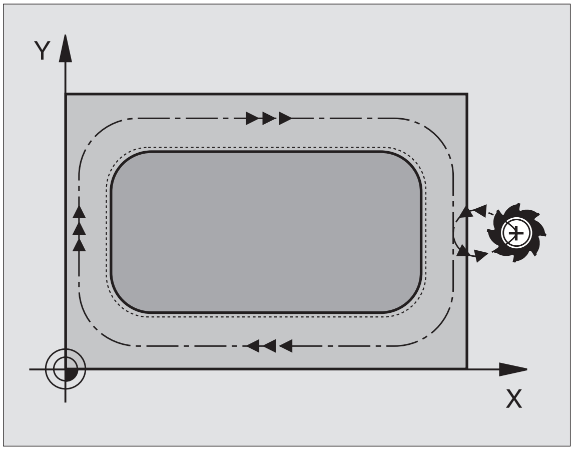
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting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

Example: NC blocks

```
N350 G212 POCKET FINISHING
     Q200=2         ;SET-UP CLEARANCE
     Q201=-20       ;DEPTH
     Q206=150       ;FEED RATE FOR PLUNGING
     Q202=5         ;PLUNGING DEPTH
     Q207=500       ;FEED RATE FOR MILLING
     Q203=+30       ;SURFACE COORDINATE
     Q204=50        ;2ND SET-UP CLEARANCE
     Q216=+50       ;CENTER IN 1ST AXIS
     Q217=+50       ;CENTER IN 2ND AXIS
     Q218=80        ;FIRST SIDE LENGTH
     Q219=60        ;SECOND SIDE LENGTH
     Q220=5         ;CORNER RADIUS
     Q221=0         ;OVERSIZE
```

STUD FINISHING (Cycle G213)

1. The TNC moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the stud.
2. From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3 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. The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6. This process (3 to 5) is repeated until the programmed depth is reached.
7. At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).

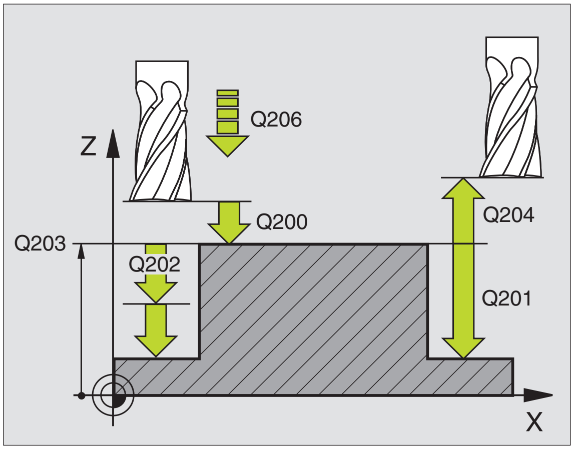


Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

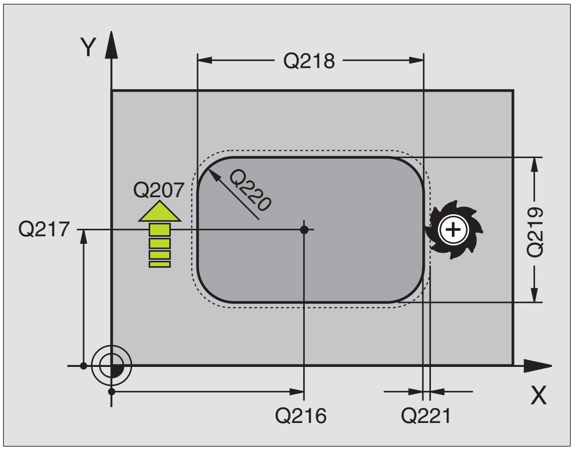
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

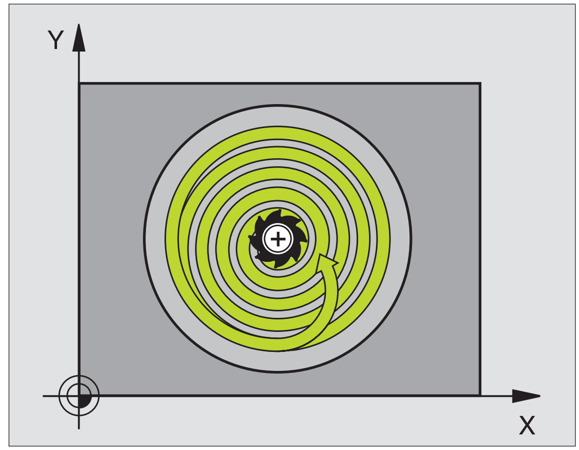


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting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

Example: NC blocks

```
N350 G213 STUD FINISHING
     Q200=2        ;SET-UP CLEARANCE
     Q291=-20      ;DEPTH
     Q206=150      ;FEED RATE FOR PLUNGING
     Q202=5        ;PLUNGING DEPTH
     Q207=500      ;FEED RATE FOR MILLING
     Q203=+30      ;SURFACE COORDINATE
     Q294=50       ;2ND SET-UP CLEARANCE
     Q216=+50      ;CENTER IN 1ST AXIS
     Q217=+50      ;CENTER IN 2ND AXIS
     Q218=80       ;FIRST SIDE LENGTH
     Q219=60       ;SECOND SIDE LENGTH
     Q220=5        ;CORNER RADIUS
     Q221=0        ;OVERSIZE
```

CIRCULAR POCKET MILLING (Cycle 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 "POCKET MILLING (Cycles G75, G76)," page 271.
3. This process is repeated until the depth is reached.
4. At the end of the cycle, the TNC retracts the tool to the starting position.

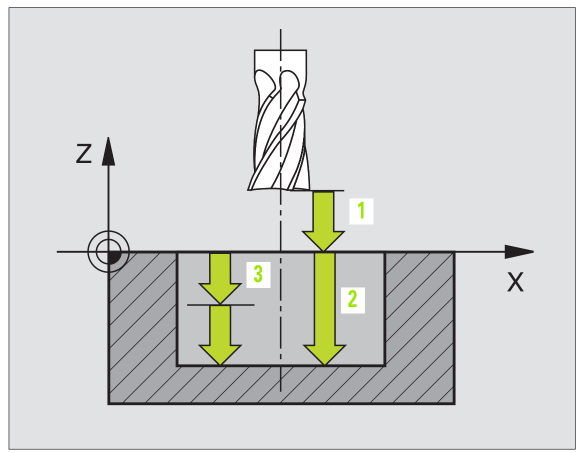
Before programming, note the following:

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

Pre-position over the pocket center with radius compensation **G40.**

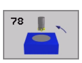
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



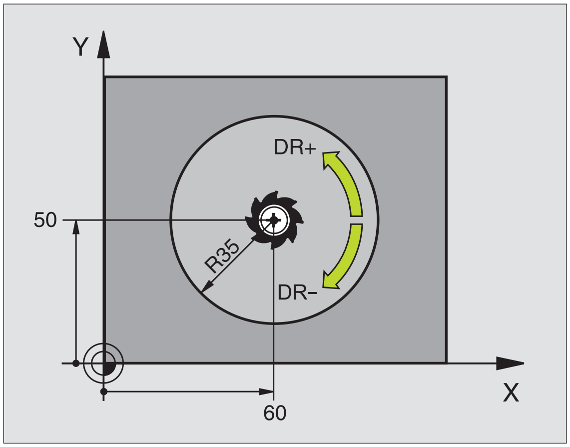
Direction of rotation during rough-out

- In clockwise direction: G77 (DR-)
- In counterclockwise direction: G78 (DR+)



- **Set-up clearance** 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **Milling depth** 2: Distance between workpiece surface and bottom of pocket.
- **Plunging depth** 3 (incremental value): Infeed per cut The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

- **`Feed rate for plunging:`** Traversing speed of the tool during penetration
- **`Circular radius:`** Radius of the circular pocket
- **`Feed rate F:`** Traversing speed of the tool in the working plane.



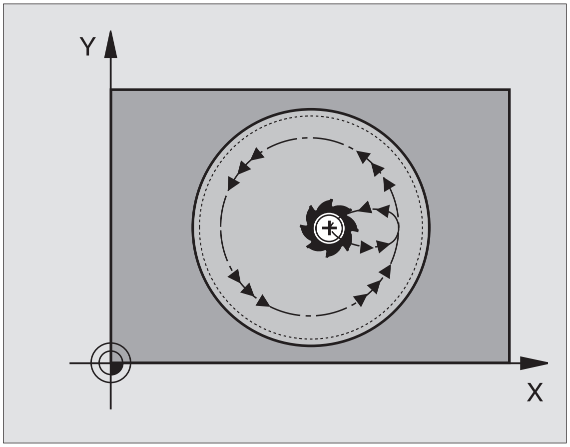
Example: NC blocks

```
N26 G77 P01 2  P02 -20  P035  P04 100
    P05 40  P06 250 *
```

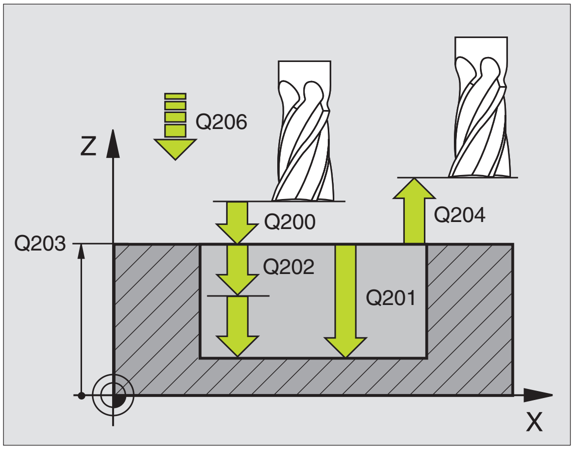
```
...
```

```
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 the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.
2. From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the workpiece blank diameter and tool radius into account for calculating the starting point. If you enter a workpiece blank diameter of 0, the TNC plunge-cuts into the pocket center.
3. If the tool is at the 2nd set-up clearance, it moves 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 at rapid traverse to set-up clearance, or, if programmed, to the 2nd set-up clearance and then to the center of the pocket (end position = starting position).

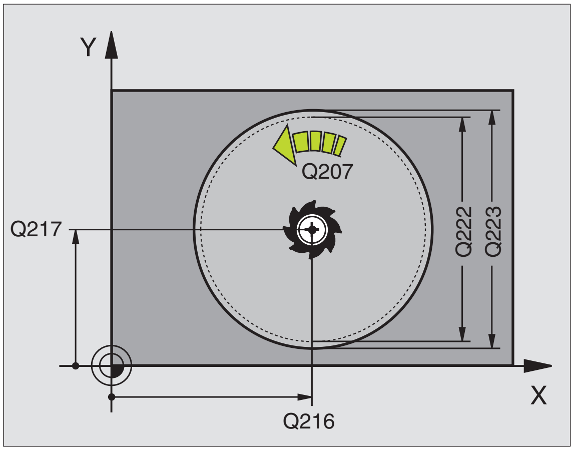


Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

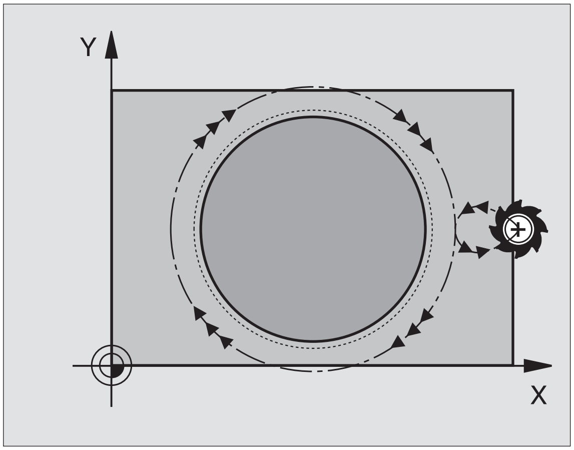
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting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

Example: NC blocks

```
N420 G214 CIRCULAR POCKET FINISHING
    Q200=2        ;SET-UP CLEARANCE
    Q201=-20      ;DEPTH
    Q206=150      ;FEED RATE FOR PLUNGING
    Q202=5        ;PLUNGING DEPTH
    Q207=500      ;FEED RATE FOR MILLING
    Q203=+30      ;SURFACE COORDINATE
    Q204=50       ;2ND SET-UP CLEARANCE
    Q216=+50      ;CENTER IN 1ST AXIS
    Q217=+50      ;CENTER IN 2ND AXIS
    Q222=79       ;WORKPIECE BLANK DIA.
    Q223=80       ;FINISHED PART DIA.
```

CIRCULAR STUD FINISHING (Cycle G215)

1 The TNC automatically moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the 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. 2 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 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 This process (3 to 5) is repeated until the programmed depth is reached.
7 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).

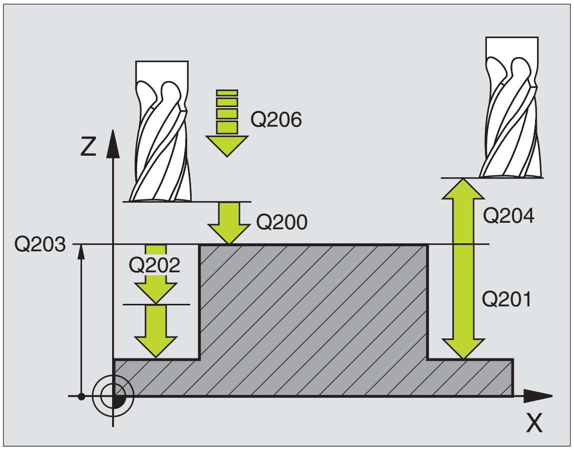


Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

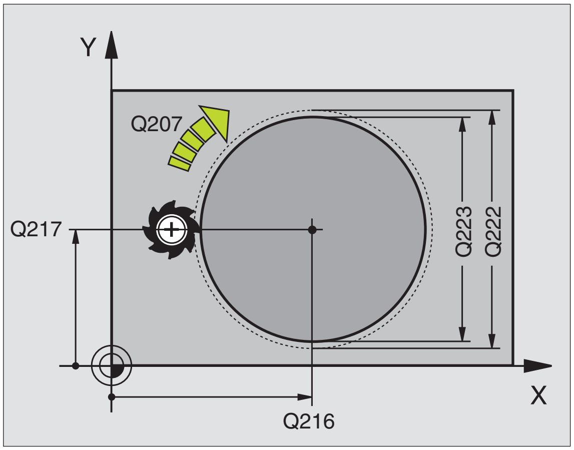
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

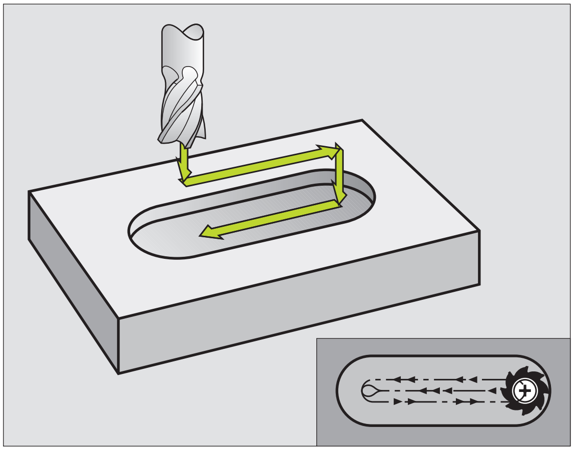


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting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

Example: NC blocks

```
N430 G215 C. STUD FINISHING
    Q200=2          ;SET-UP CLEARANCE
    Q201=-20        ;DEPTH
    Q206=150        ;FEED RATE FOR PLUNGING
    Q202=5          ;PLUNGING DEPTH
    Q207=500        ;FEED RATE FOR MILLING
    Q203=+30        ;SURFACE COORDINATE
    Q204=50         ;2ND SET-UP CLEARANCE
    Q216=+50        ;CENTER IN 1ST AXIS
    Q217=+50        ;CENTER IN 2ND AXIS
    Q222=81         ;WORKPIECE BLANK DIA.
    Q223=80         ;FINISHED PART DIA.
```

SLOT MILLING (Cycle 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 at rapid traverse to the 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:

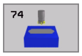
This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the starting point.

Pre-position to the center of the slot and offset by the tool radius into the slot with radius compensation **G40.**

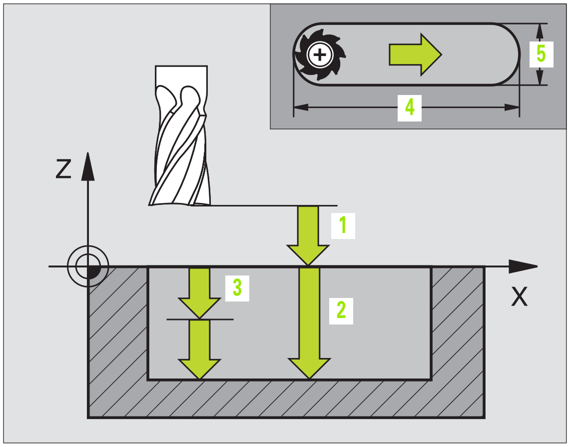
The cutter diameter must be not be larger than the slot width and not smaller than half the slot width.

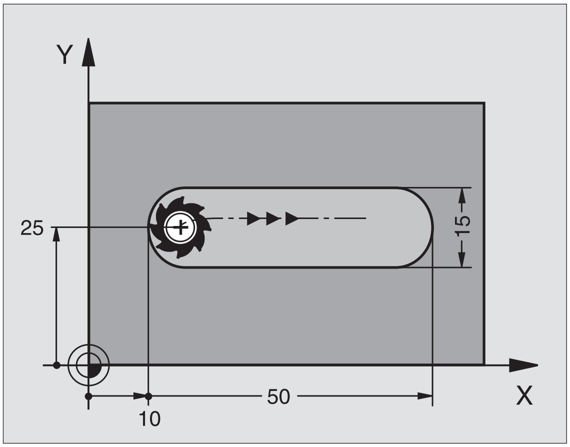
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **Milling depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut. The tool will drill to the depth in one operation if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Feed rate for plunging:** Traversing speed during penetration.
- **1st side length 4**: 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  P0  5  P04 100
    P05 X+80  P06 Y+12 P07 275 *
```

SLOT with reciprocating plunge-cut (Cycle G210)

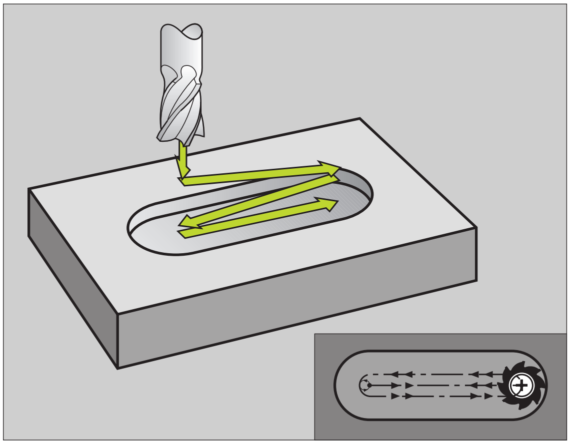
Roughing process

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 the set-up clearance above the workpiece surface.

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

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

4 For the purpose of face milling, the TNC moves the tool at the milling depth to the other end of the slot and then back to the center of the slot.

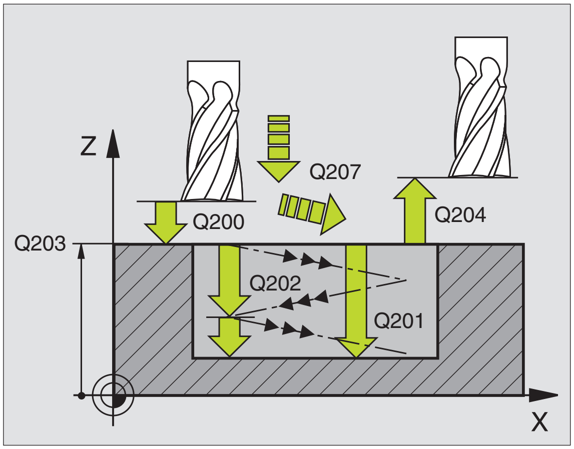


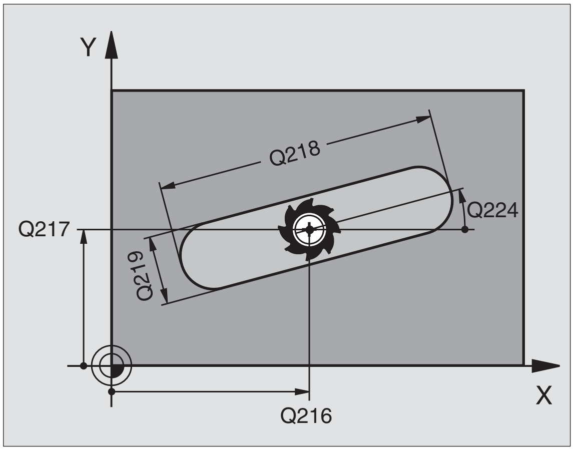
Finishing process

5 The TNC positions the tool in the center of the left circle and then moves it tangentially to the left end of the slot. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed.

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

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





Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

During roughing the tool plunges into the material with a sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

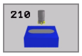
The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

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

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



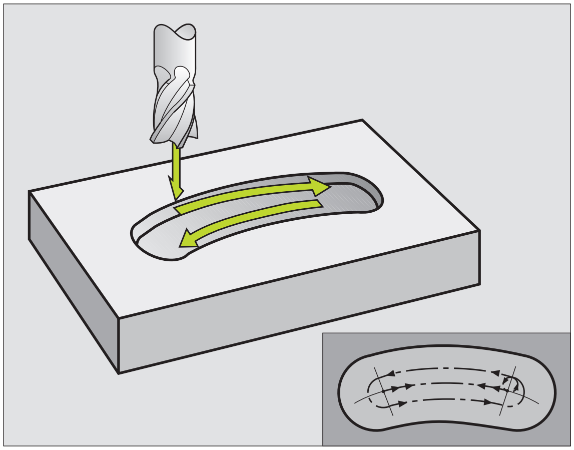
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Plunging depth** Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement.
- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the slot in the reference axis of the working plane.
- **Center in 2nd axis** Q217 (absolute value): Center of the slot in the minor axis of the working plane.
- **First side length** Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot.
- **Second side length** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).
- **Angle of rotation** Q224 (absolute value): Angle by which the entire slot is rotated. The center of rotation lies in the center of the slot.
- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Effective only during finishing if infeed for finishing is entered.

Example: NC blocks

```
N510 G210 SLOT RECIP. PLNG
    Q200=2          ;SET-UP CLEARANCE
    Q201=-20        ;DEPTH
    Q207=500        ;FEED RATE FOR MILLING
    Q202=5          ;PLUNGING DEPTH
    Q215=0          ;MACHINING OPERATION
    Q203=+30        ;SURFACE COORDINATE
    Q204=50         ;2ND SET-UP CLEARANCE
    Q216=+50        ;CENTER IN 1ST AXIS
    Q217=+50        ;CENTER IN 2ND AXIS
    Q218=80         ;FIRST SIDE LENGTH
    Q219=12         ;SECOND SIDE LENGTH
    Q224=+15        ;ANGLE OF ROTATION
    Q338=5          ;INFEED FOR FINISHING
    Q206=150        ;FEED RATE FOR PLUNGING
```

CIRCULAR SLOT 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 (steps 2 to 3) is repeated until the programmed milling depth is reached.

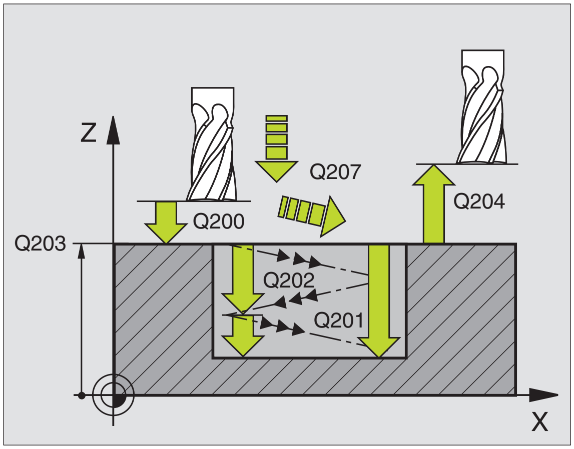
4 For the purpose of face milling, the TNC moves the tool at the milling depth to the other end of the slot.

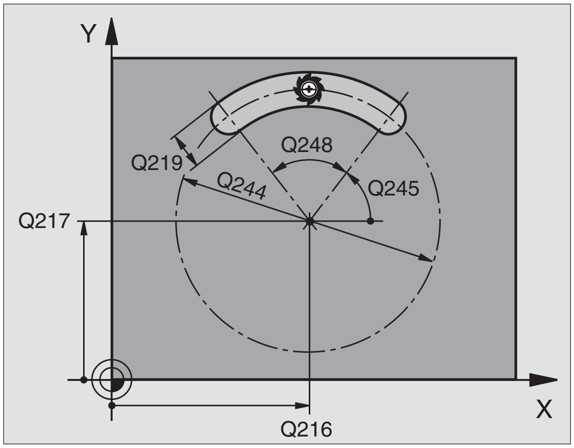
Finishing process

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), and if so entered, in more than one infeed. The starting point for the finishing process is the center of the right circle.

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

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





Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

During roughing the tool plunges into the material with a helical sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

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

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

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

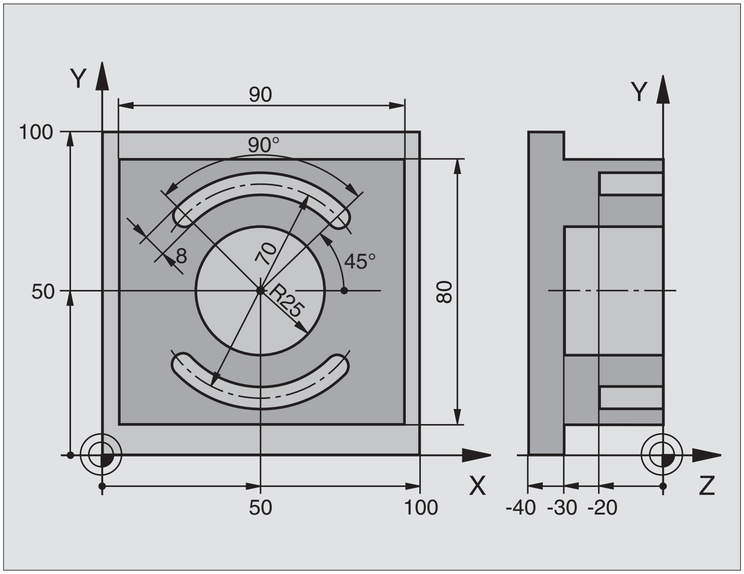
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Plunging depth** Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement.
- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the slot in the reference axis of the working plane.
- **Center in 2nd axis** Q217 (absolute value): Center of the slot in the minor axis of the working plane.
- **Pitch circle diameter** Q244: Enter the diameter of the pitch circle.
- **Second side length** Q219: Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).
- **Starting angle** Q245 (absolute value): Enter the polar angle of the starting point.
- **Angular length** Q248 (incremental value): Enter the angular length of the slot.
- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Effective only during finishing if infeed for finishing is entered.

Example: NC blocks

```
N520 G211 CIRCULAR SLOT
     Q200=2         ;SET-UP CLEARANCE
     Q201=-20       ;DEPTH
     Q207=500       ;FEED RATE FOR MILLING
     Q202=5         ;PLUNGING DEPTH
     Q215=0         ;MACHINING OPERATION
     Q203=+30       ;SURFACE COORDINATE
     Q204=50        ;2ND SET-UP CLEARANCE
     Q216=+50       ;CENTER IN 1ST AXIS
     Q217=+50       ;CENTER IN 2ND AXIS
     Q244=80        ;PITCH CIRCLE DIA.
     Q219=12        ;SECOND SIDE LENGTH
     Q245=+45       ;STARTING ANGLE
     Q248=90        ;ANGULAR LENGTH
     Q338=5         ;INFEED FOR FINISHING
     Q206=150       ;FEED RATE FOR PLUNGING
```

Example: Milling pockets, studs and slots



%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 the tool for roughing/finishing
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G213 STUD FINISHING	Define cycle for machining the contour outside
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q207=250 ;FEED RATE FOR MILLING	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=90 ;FIRST SIDE LENGTH	
Q219=80 ;SECOND SIDE LENGTH	
Q220=0 ;CORNER RADIUS	
Q221=5 ;OVERSIZE	

`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 CIRCULAR SLOT`	Cycle definition for slot 1
`Q200=2 ;SET-UP CLEARANCE`	
`Q201=-20 ;DEPTH`	
`Q207=250 ;FEED RATE FOR MILLING`	
`Q202=5 ;PLUNGING DEPTH`	
`Q215=0 ;MACHINING OPERATION`	
`Q203=+0 ;SURFACE COORDINATE`	
`Q204=100 ;2ND SET-UP CLEARANCE`	
`Q216=+50 ;CENTER IN 1ST AXIS`	
`Q217=+50 ;CENTER IN 2ND AXIS`	
`Q244=70 ;PITCH CIRCLE DIA.`	
`Q219=8 ;SECOND SIDE LENGTH`	
`Q245=+45 ;STARTING ANGLE`	
`Q248=90 ;ANGULAR LENGTH`	
`Q338=5 ;INFEED FOR FINISHING`	
`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

Overview

The TNC provides two cycles for machining hole patterns directly:

Cycle	Soft key
G220 CIRCULAR PATTERN	
G221 LINEAR PATTERN	

You can combine Cycle G220 and Cycle G221 with the following fixed cycles:

> If you have to machine irregular hole patterns, use **G79 "PAT"** to develop point tables (see "Point Tables" on page 218).

Cycle G74	SLOT MILLING
Cycle G75/G76	POCKET MILLING
Cycle G77/G78	CIRCULAR POCKET MILLING
Cycle G83	PECKING
Cycle G84	TAPPING with a floating tap holder
Cycle G85	RIGID TAPPING without a floating tap holder
Cycle G86	THREAD CUTTING
Cycle G200	DRILLING
Cycle G201	REAMING
Cycle G202	BORING
Cycle G203	UNIVERSAL DRILLING
Cycle G204	BACK BORING
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
Cycle G209	TAPPING WITH CHIP BREAKING
Cycle G212	POCKET FINISHING
Cycle G213	STUD FINISHING
Cycle G214	CIRCULAR POCKET FINISHING
Cycle G215	CIRCULAR STUD FINISHING
Cycle G262	THREAD MILLING
Cycle G263	THREAD MILLING/COUNTERSINKING
Cycle G264	THREAD DRILLING/MILLING
Cycle G265	HELICAL THREAD DRILLING/MILLING
Cycle G267	OUTSIDE THREAD MILLING

CIRCULAR PATTERN (Cycle G220)

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

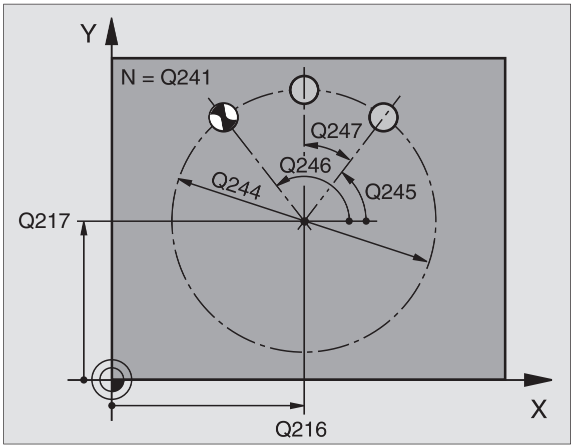
Sequence:

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

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

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

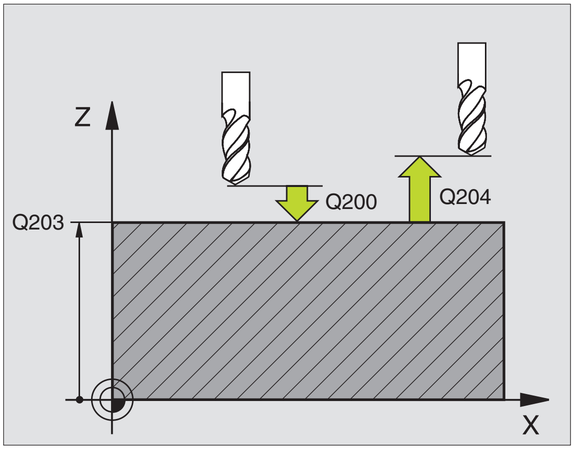
4 This process (1 to 3) is repeated until all machining operations have been executed.



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 G209, G212 to G215 and G262 to G267, 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 reference axis of the working plane.
- **`Center in 2nd axis`** Q217 (absolute value): Center of the pitch circle in the minor axis of the working plane.
- **`Pitch circle diameter`** Q244: Diameter of the pitch circle.
- **`Starting angle`** Q245 (absolute value): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle.
- **`Stopping angle`** Q246 (absolute value): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise.

Example: NC blocks

```
N530 G220 POLAR PATTERN
     Q216=+50       ;CENTER IN 1ST AXIS
     Q217=+50       ;CENTER IN 2ND AXIS
     Q244=80        ;PITCH CIRCLE DIA.
     Q245=+0        ;STARTING ANGLE
     Q246=+360      ;STOPPING ANGLE
     Q247=+0        ;STEPPING ANGLE
     Q241=8         ;NR OF REPETITIONS
     Q200=2         ;SET-UP CLEARANCE
     Q203=+30       ;SURFACE COORDINATE
     Q204=50        ;2ND SET-UP CLEARANCE
     Q203=1         ;MOVE TO CLEARANCE
```

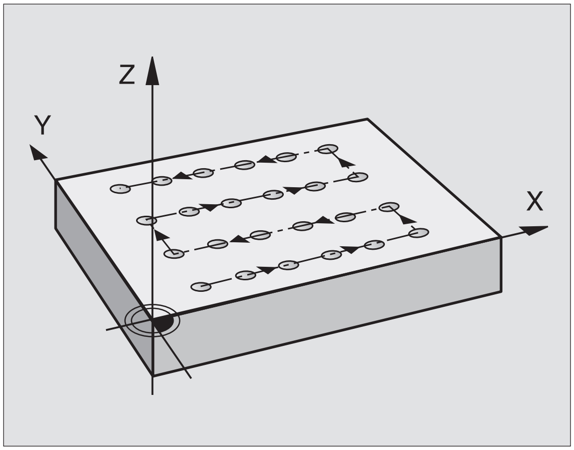
- **Stepping angle** Q247 (incremental value): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will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

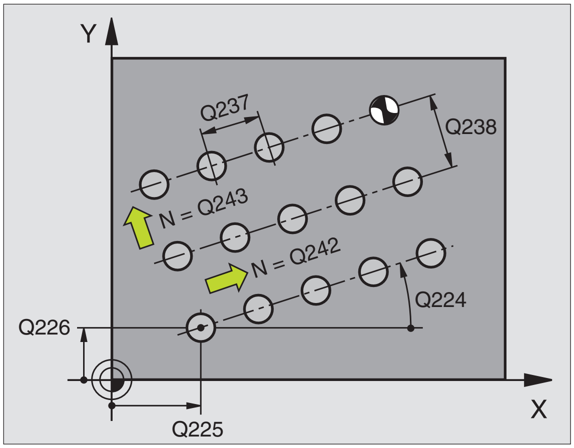
LINEAR PATTERN (Cycle G221)

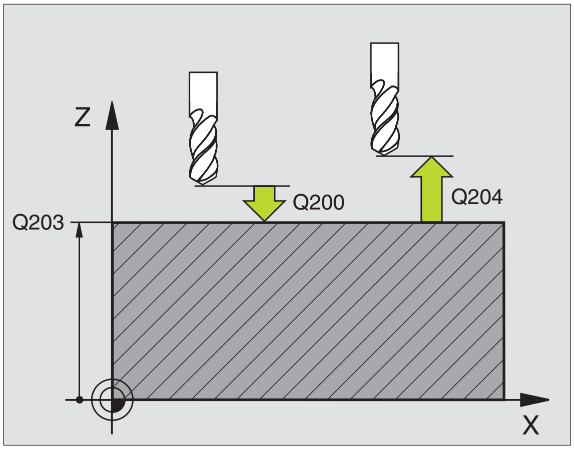
Before programming, note the following:

Cycle G221 is DEF active, which means that Cycle G221 automatically calls the last defined fixed cycle.

If you combine Cycle G221 with one of the fixed cycles G200 to G209, G212 to G215 and G262 to G267, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle G221 will be effective for the selected fixed cycle.







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

Sequence:

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

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

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

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

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

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

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

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

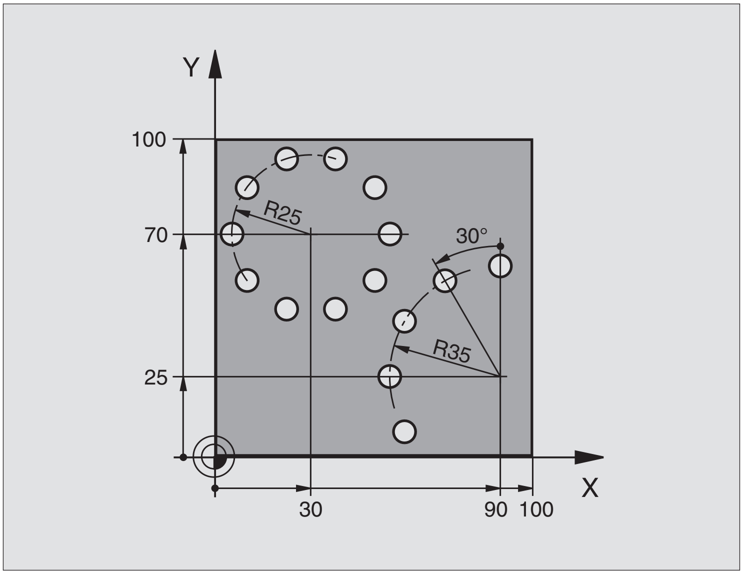
9 All subsequent lines are processed in a reciprocating movement.

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

Example: NC blocks

```
N540 G221 CARTESIAN PATTERN
     Q225=+15        ;STARTING PNT 1ST AXIS
     Q226=+15        ;STARTING PNT 2ND AXIS
     Q237=+10        ;SPACING IN 1ST AXIS
     Q238=+8         ;SPACING IN 2ND AXIS
     Q242=6          ;NUMBER OF COLUMNS
     Q243=4          ;NUMBER OF LINES
     Q224=+15        ;ANGLE OF ROTATION
     Q200=2          ;SET-UP CLEARANCE
     Q203=+30        ;SURFACE COORDINATE
     Q204=50         ;2ND SET-UP CLEARANCE
     Q301=1          ;MOVE TO CLEARANCE
```

Example: Circular hole patterns



%PATTERN 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 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=4 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME	
Q203=+0 ;SURFACE COORDINATE	
Q204=0 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	

N70 G220 POLAR PATTERN	Define cycle for circular pattern 1, CYCL 200 is called automatically,
Q216=+30 ;CENTER IN 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+70 ;CENTER IN 2ND AXIS	
Q244=50 ;PITCH CIRCLE DIA.	
Q245=+0 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+0 ;STEPPING ANGLE	
Q241=10 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
N80 G220 POLAR PATTERN	Define cycle for circular pattern 2, CYCL 200 is called automatically,
Q216=+90 ;CENTER IN 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+25 ;CENTER IN 2ND AXIS	
Q244=70 ;PITCH CIRCLE DIA.	
Q245=+90 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=30 ;STEPPING ANGLE	
Q241=5 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
N90 G00 G40 Z+250 M02 *	Retract in the tool axis, end program
N999999 %PATTERN G71	

8.6 SL Cycles Group I

Fundamentals

SL Cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle **G37** CONTOUR GEOMETRY.

> The memory capacity for programming an SL cycle (all contour subprograms) is limited to 48 kilobytes. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. For example, you can program up to approx. 256 line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation **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

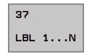
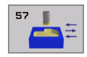
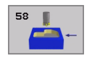
- The TNC automatically positions the tool to the starting position in the machining plane before a cycle. In the spindle axis the tool must be pre-positioned to set-up clearance.
- Each level of infeed depth is roughed-out axis-parallel or at a preset angle (angle defined in Cycle **G57**). In the standard 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.

> With MP7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

Example: Program structure: Machining with SL Cycles

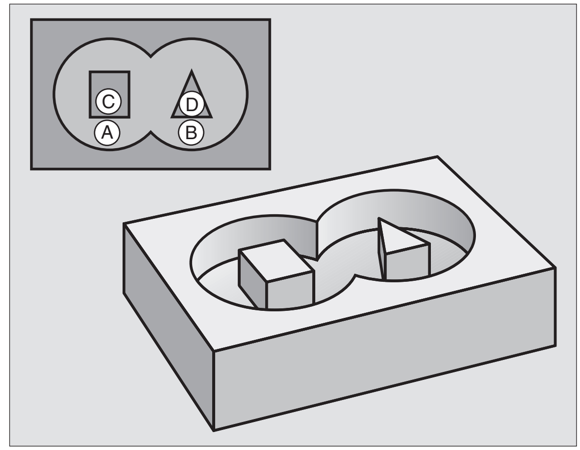
```
%SL G71 *
...
N12 G37 P01 ...
...
N16 G56 P01 ...
N17 G79 *
...
N18 G57 P01 ...
N19 G79 *
...
N26 G59 P01 ...
N27 G79 *
...
N50 G00 G40 G90 Z+250 M2 *
N51 G98 L1 *
...
N60 G98 L0 *
N61 G98 L2 *
...
N62 G98 L0 *
...
N999999 %SL G71 *
```

Overview of SL Cycles, Group I

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.



Before programming, note the following:

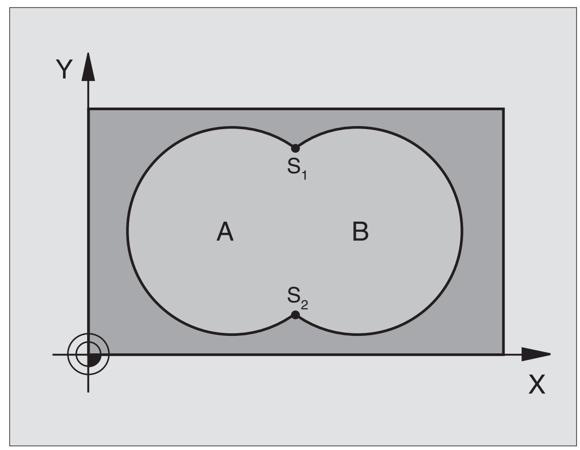
Cycle **G37** is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can list up to 12 subroutines (subcontours) in Cycle **G37.**

37
LBL 1...N

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

Overlapping contours: (see "Overlapping contours" on page 308)



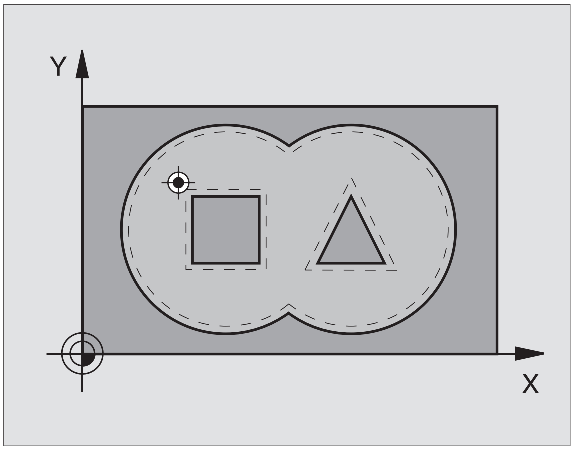
Example: NC blocks

```
N54 G37 P01 1  P02 5  P03 7  P04  8 *
```

PILOT DRILLING (Cycle G56)

Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

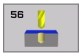


Process

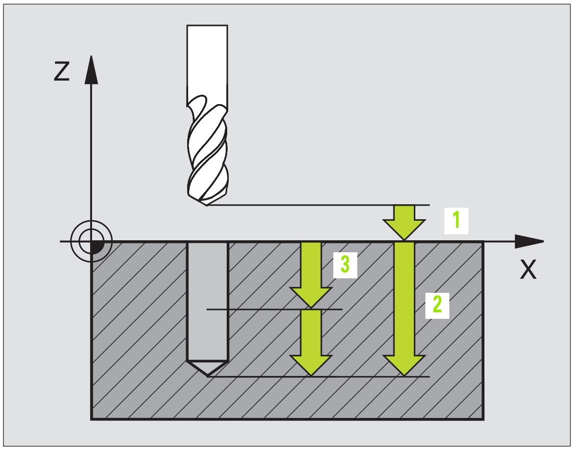
Same as Cycle **G83** Pecking (see "Cycles for Drilling, Tapping and Thread Milling," page 222).

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.



- **Set-up clearance** 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **Total hole depth** 2 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **Plunging depth** 3 (incremental value): Infeed per cut The total hole depth does not have to be a multiple of the plunging depth. The tool will drill to the total hole depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the total hole depth
- **Feed rate for plunging**: Traversing speed in mm/min for drilling
- **Finishing allowance:** Allowance in the machining plane



Example: NC blocks

```
N54 G56 P01 2  P02 -15  P03 5  P04 250
    P05 +0.5*
```

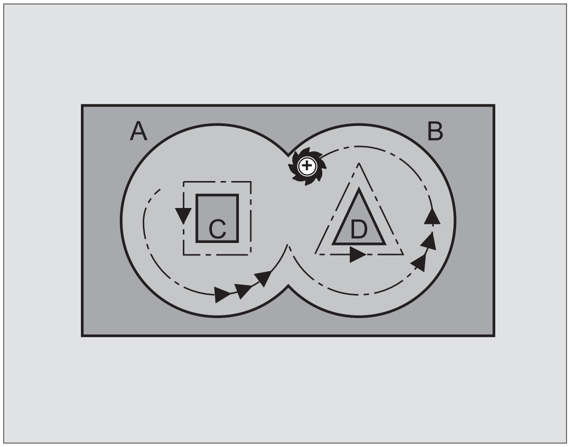
ROUGH-OUT (Cycle G57)

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.

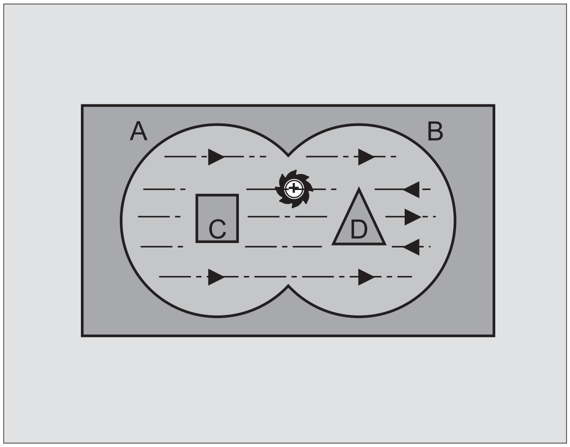
The contour is fully rough-milled (see figure at top right):

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 programmed milling depth is reached.



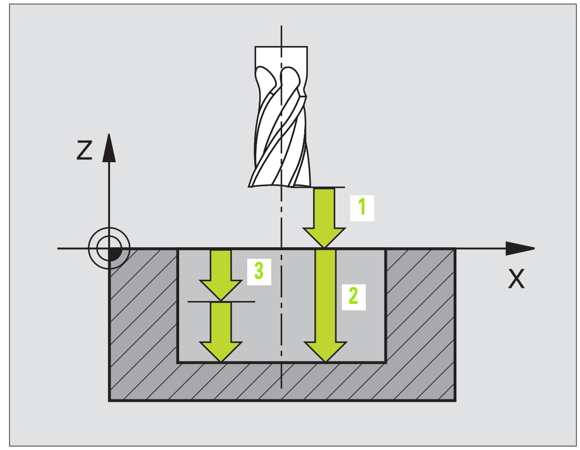
Before programming, note the following:

With MP7420.0 and MP7420.1 you define how the TNC should machine the contour (see “General User Parameters” on page 466).

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 21.

- **`Set-up clearance`** 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **`Milling depth`** 2 (incremental value): Distance between workpiece surface and bottom of pocket
- **`Plunging depth`** 3 (incremental value): Infeed per cut The milling depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - The plunging depth is greater than the milling 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 reference 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)

Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

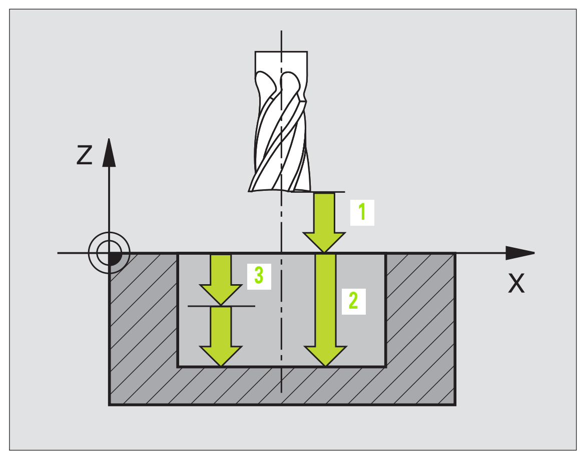
Application

Cycle G58/G59 CONTOUR MILLING serves for finishing the contour pocket.

Direction of rotation during contour milling

- In clockwise direction: **`G58`**
- In counterclockwise direction: **`G59`**

- **`Set-up clearance`** 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **`Milling depth`** 2 (incremental value): Distance between workpiece surface and bottom of pocket
- **`Plunging depth`** 3 (incremental value): Infeed per cut The milling depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - The plunging depth is greater than the milling 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

Fundamentals

SL Cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle **G37** CONTOUR GEOMETRY.

> The memory capacity for programming an SL cycle (all contour subprograms) is limited. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. For example, you can program up to approx. 1024 line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation **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 the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of “inside corners” can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

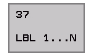
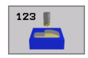
> With MP7420 you can determine where the tool is positioned at the end of Cycles G121 to G124.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle **G120.**

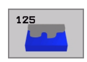
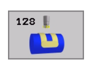
Example: Program structure: Machining with SL Cycles

```
%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 *
```

Overview of SL Cycles

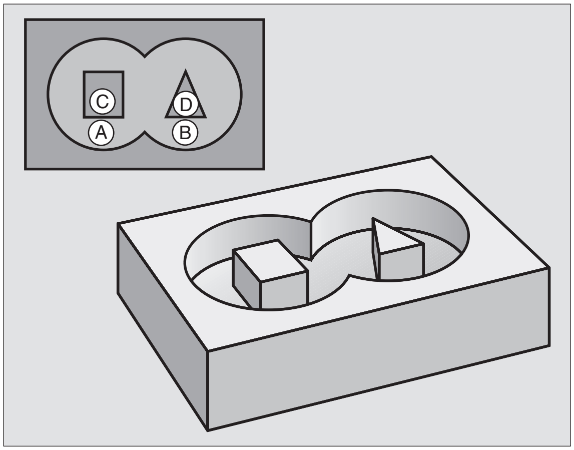
Cycle	Soft key
G37 CONTOUR GEOMETRY (essential)	
G120 CONTOUR DATA (essential)	
G121 PILOT DRILLING (optional)	
G122 ROUGH-OUT (essential)	
G123 FLOOR FINISHING (optional)	
G124 SIDE FINISHING (optional)	

Enhanced cycles:

Cycle	Soft key
G125 CONTOUR TRAIN	
G127 CYLINDER SURFACE	
G128 CYLINDER SURFACE slot milling	

CONTOUR GEOMETRY (Cycle G37)

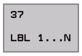
All subprograms that are superimposed to define the contour are listed in Cycle **G37** CONTOUR GEOMETRY.



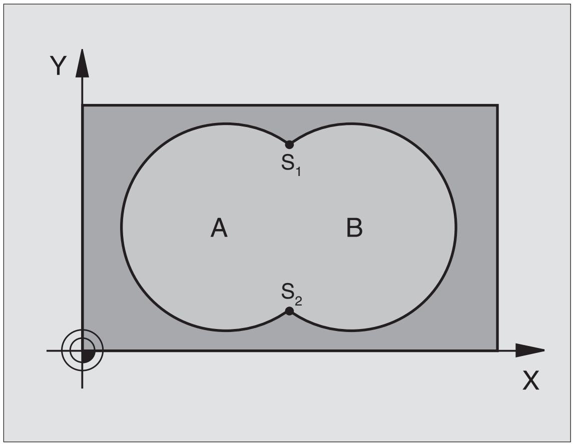
Before programming, note the following:

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 subroutines (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 blocks

```
N120 G37 P01 1  P02 5  P03 7  P04 8 *
```

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 S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Subprogram 1: Pocket 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 *
```

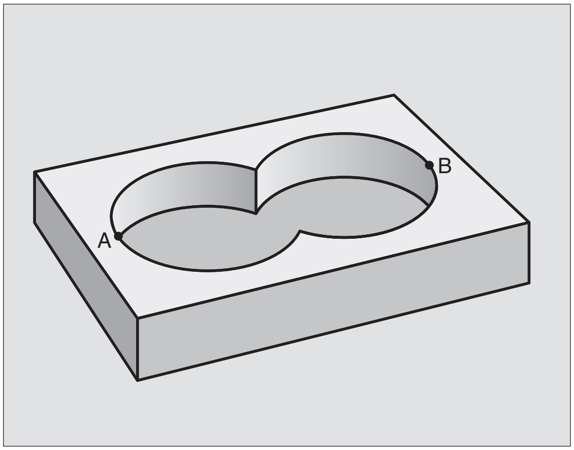
Subprogram 2: Pocket B

```
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 inclusion

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

- The surfaces A and B must be pockets.
- The first pocket (in Cycle **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 *
```

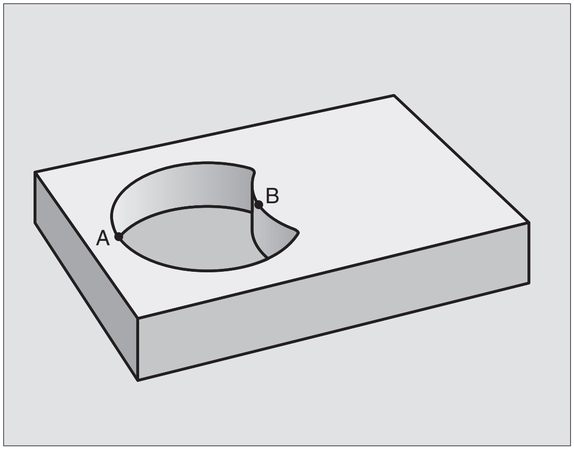
Area of exclusion

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 *
```

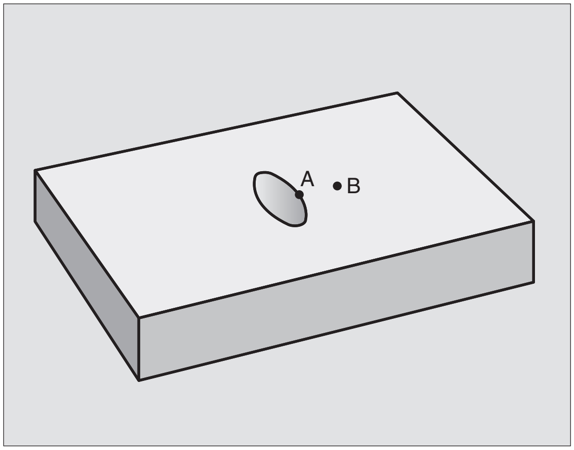
Area of intersection

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

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

Surface A:

```
N510 G98 L1 *
N520 G01 G42 X+60 Y+50 *
N530 I+35 J+50 *
N540 G02 X+60 Y+50 *
N550 G98 L0 *
```

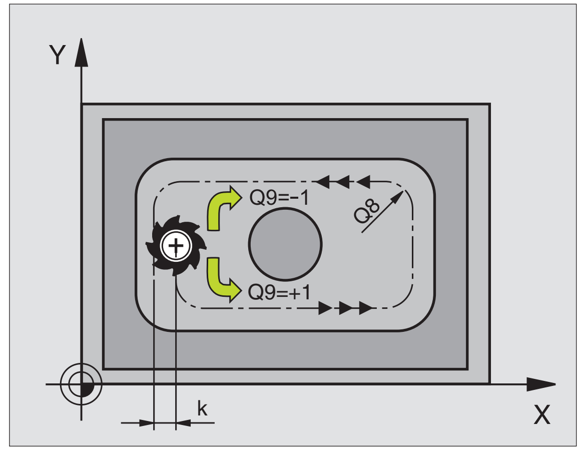


Surface B:

```
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.**



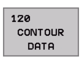
Before programming, note the following:

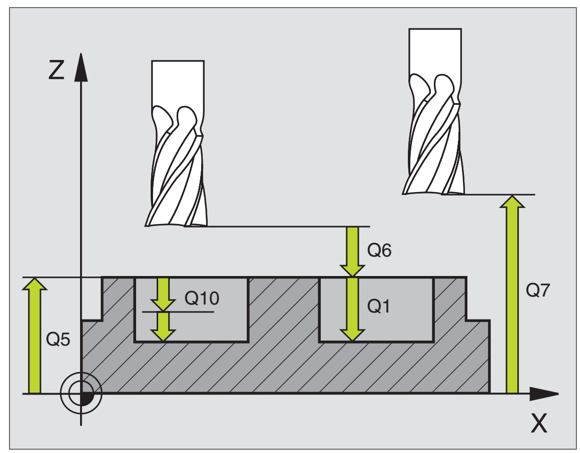
Cycle **G120** is DEF active which means that Cycle **G120** becomes effective as soon as it is defined in the part program.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the TNC does not execute that next cycle.

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.





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

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

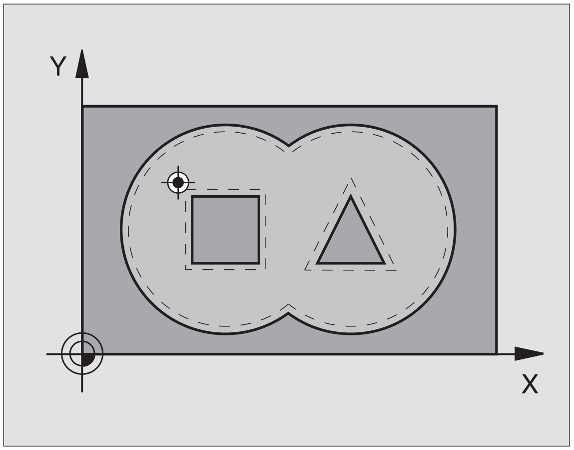
Example: NC block

```
N57 G120 CONTOUR DATA
    Q1=-20          ;MILLING DEPTH
    Q2=1            ;TOOL PATH OVERLAP
    Q3=+0.2         ;ALLOWANCE FOR SIDE
    Q4=+0.1         ;ALLOWANCE FOR FLOOR
    Q5=+30          ;SURFACE COORDINATE
    Q6=2            ;SET-UP CLEARANCE
    Q7=+80          ;CLEARANCE HEIGTH
    Q8=0.5          ;ROUNDING RADIUS
    Q9=+1           ;DIRECTION OF ROTATION
```

PILOT DRILLING (Cycle G121)

When calculating the infeed points, the TNC does not account for the delta value **DR** programmed in a **T** block.

In narrow areas, the TNC may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

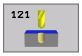


Process

Same as Cycle **G83** Pecking (see “Cycles for Drilling, Tapping and Thread Milling,” page 222).

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.



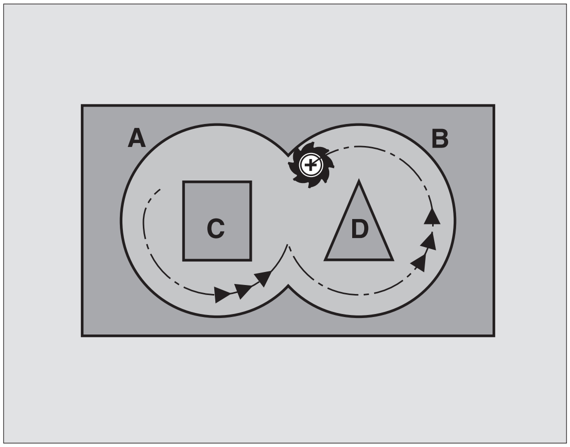
- **Plunging depth** Q10 (incremental value): Dimension by which the tool drills in each infeed (negative sign for negative working direction).
- **Feed rate for plunging** Q11: Traversing speed in mm/min during drilling.
- **Rough-out tool number** Q13: Tool number of the roughing mill.

Example: NC blocks

```
N58 G121 PILOT DRILLING
    Q10=+5          ;PLUNGING DEPTH
    Q11=100         ;FEED RATE FOR PLUNGING
    Q13=1           ;ROUGH-OUT TOOL
```

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 the inside outward at the milling feed rate Q12.
3 The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B).
4 Then the TNC rough-mills the pocket contour and retracts the tool to the clearance height.



Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle **G121.**

If you define a plunge angle in the ANGLE column of the tool table for the roughing tool, the TNC moves on a helical path to the respective roughing depth (see “Tool table: Standard tool data” on page 133).

122

- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool in mm/min during penetration.
- **Feed rate for milling** Q12: Traversing speed for milling in mm/min.
- **Coarse roughing tool number** Q18: Number of the tool with which the TNC has already coarse-roughed the contour. If there was no coarse roughing, enter “0”; if you enter a value other than zero, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool.
  If the portion that is to be roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut; For this purpose you must enter the tool length LCUTS in the tool table TOOL.T (see “Tool Data,” page 131) and define the maximum plunging ANGLE of the tool. The TNC will otherwise generate an error message.
- **Reciprocation feed rate** Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut.

Example: NC block

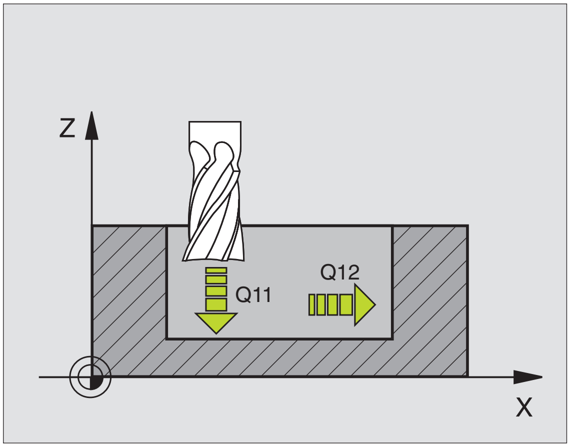
```
N59 G122 ROUGH-OUT
    Q10=+5         ;PLUNGING DEPTH
    Q11=100        ;FEED RATE FOR PLUNGING
    Q12=350        ;FEED RATE FOR ROUGHING
    Q18=1          ;COARSE ROUGHING TOOL
    Q19=150        ;RECIPROCATION FEED RATE
```

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** Q11: Traversing speed of the tool during penetration.
- **Feed rate for milling** Q12: Traversing speed for milling.



Example: NC block

```
N60 G123 FLOOR FINISHING
    Q11=100        ;FEED RATE FOR PLUNGING
    Q12=350        ;FEED RATE FOR ROUGHING
```

SIDE FINISHING (Cycle G124)

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

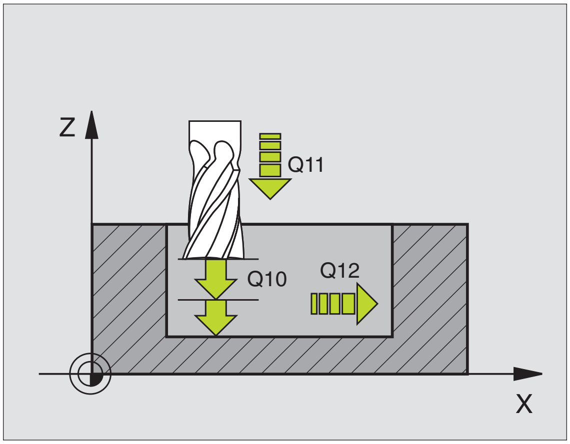
Before programming, note the following:

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: Machining direction:
  +1: Counterclockwise
  -1: Clockwise
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration.
- **Feed rate for milling** Q12: Traversing speed for milling.
- **Finishing allowance for side** Q14 (incremental value): Enter the allowed material for several finish-milling operations. If you enter Q14 = 0, the remaining finishing allowance will be cleared.

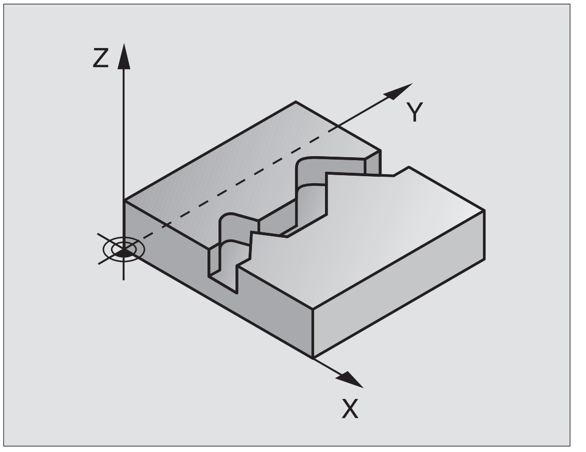


Example: NC block

```
N61 G124 SIDE FINISHING
    Q9=+1          ;DIRECTION OF ROTATION
    Q10=+5         ;PLUNGING DEPTH
    Q11=100        ;FEED RATE FOR PLUNGING
    Q12=350        ;FEED RATE FOR ROUGHING
    Q14=+0         ;ALLOWANCE FOR SIDE
```

CONTOUR TRAIN (Cycle G125)

In conjunction with Cycle **G37** CONTOUR GEOMETRY, this cycle facilitates the machining of open contours (i.e. where the starting point of the contour is not the same as its end point).



Cycle **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 cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

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 1024 straight-line blocks in one SL cycle.

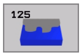
Cycle **G120** CONTOUR DATA is not required.

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.

Danger of collision!

To avoid collisions,

- Do not program positions in incremental dimensions immediately after Cycle **G125,** since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.



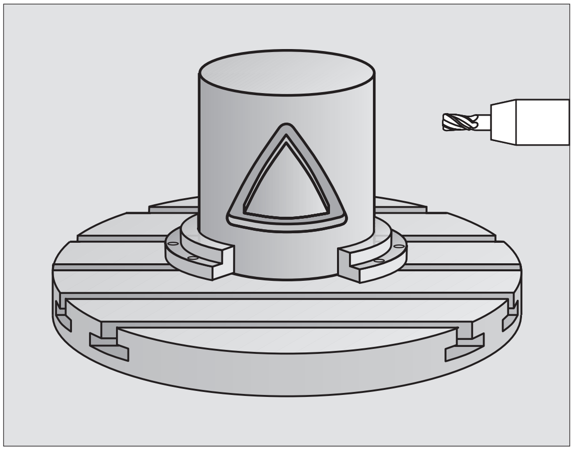
- **`Milling depth`** Q1 (incremental value): Distance between workpiece surface and contour floor.
- **`Finishing allowance for side`** Q3 (incremental value): Finishing allowance in the working plane.
- **`Workpiece surface coordinate`** Q5 (absolute value): Absolute coordinate of the workpiece surface 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
  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 CONTOUR TRAIN
    Q1=-20         ;MILLING DEPTH
    Q3=+0          ;ALLOWANCE FOR SIDE
    Q5=+0          ;SURFACE COORDINATE
    Q7=+50         ;CLEARANCE HEIGTH
    Q10=+5         ;PLUNGING DEPTH
    Q11=100        ;FEED RATE FOR PLUNGING
    Q12=350        ;FEED RATE FOR MILLING
    Q15=-1         ;CLIMB OR UP-CUT
```

CYLINDER SURFACE (Cycle G127)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

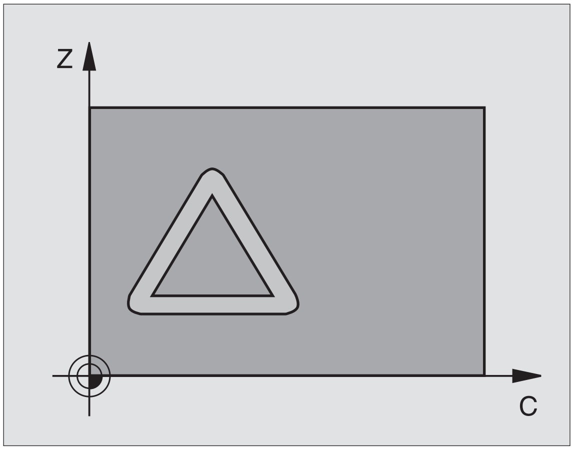


This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle **G128** if you wish to mill guide notches onto the cylinder surface.

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 set-up clearance.

Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 1024 straight-line blocks in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

The TNC checks whether the compensated and non-compensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message “Contour programming error” is output, set MP 810.x = 0.

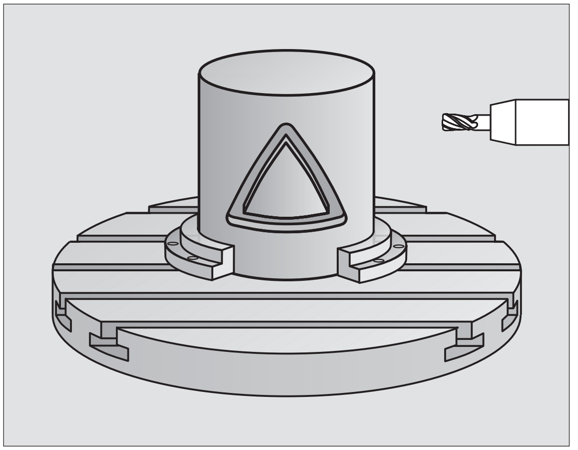
- **Milling depth** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius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

Example: NC block

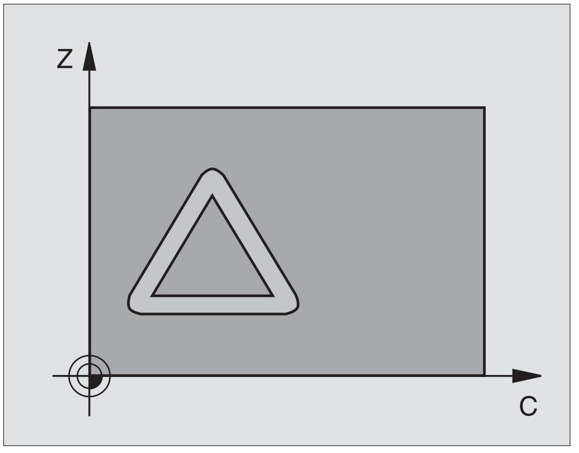
```
N63 G127 CYLINDER SURFACE
    Q1=-8          ;MILLING DEPTH
    Q3=+0          ;ALLOWANCE FOR SIDE
    Q6=+0          ;SET-UP CLEARANCE
    Q10=+3         ;PLUNGING DEPTH
    Q11=100        ;FEED RATE FOR PLUNGING
    Q12=350        ;FEED RATE FOR MILLING
    Q16=25         ;RADIUS
    Q17=0          ;DIMENSION TYPE
```

CYLINDER SURFACE slot milling (Cycle G128)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.



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 midpoint path of the contour together with the tool radius compensation. With the radius compensation you specify whether the TNC cuts the slot with climb milling or up-cut milling:



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 and 3 are repeated until the programmed milling depth Q1 is reached.
5. Then the tool moves to the set-up clearance.

Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 1024 straight-line blocks in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

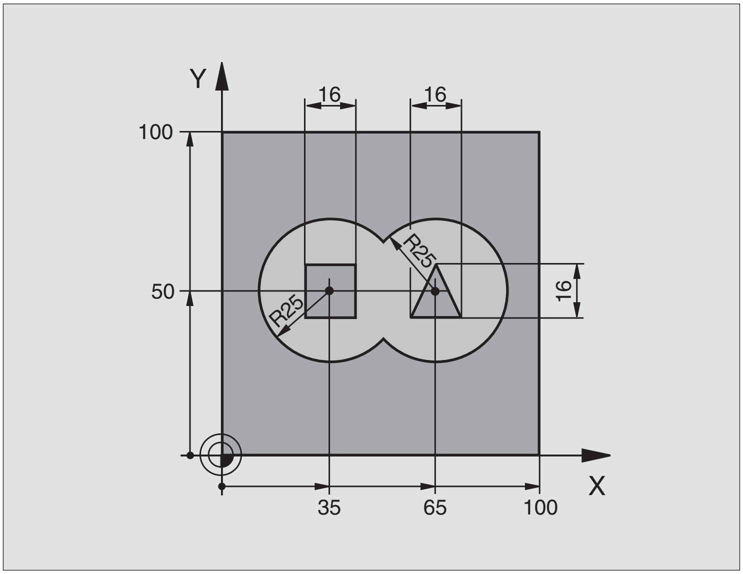
The TNC checks whether the compensated and non-compensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message “Contour programming error” is output, set MP 810.x = 0.

- **Milling depth** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation.
- **Set-up clearance** Q6 (incremental value): Distance between the tool tip and the cylinder surface.
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool in the tool axis.
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane.
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined.
- **Dimension type ? ang./lin.** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).
- **Slot width** Q20: Width of the slot to be machined.

Example: NC block

```
N63 G128 CYLINDER SURFACE
    Q1=-8          ;MILLING DEPTH
    Q3=+0          ;ALLOWANCE FOR SIDE
    Q6=+0          ;SET-UP CLEARANCE
    Q10=+3         ;PLUNGING DEPTH
    Q11=100        ;FEED RATE FOR PLUNGING
    Q12=350        ;FEED RATE FOR MILLING
    Q16=25         ;RADIUS
    Q17=0          ;DIMENSION TYPE
    Q20=12         ;SLOT WIDTH
```

Example: Pilot drilling, roughing-out and finishing overlapping contours

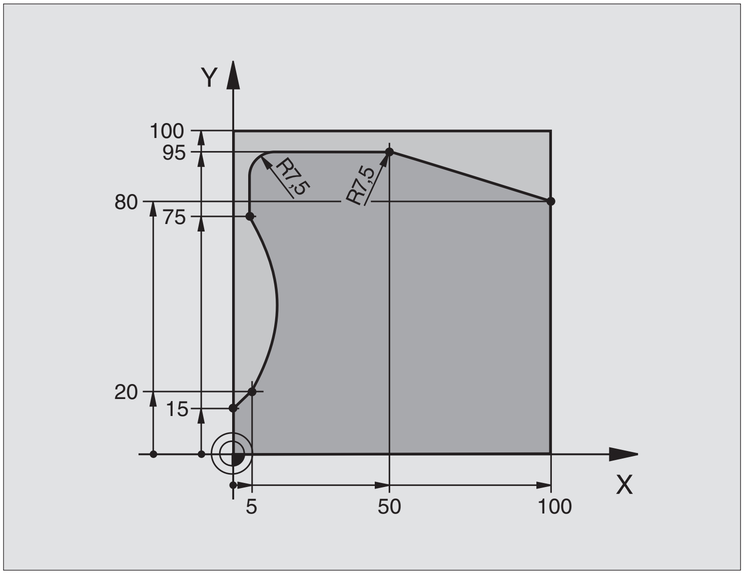


`%C21 G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+6 *`	Define tool: drill
`N40 G99 T2 L+0 R+6 *`	Define the tool for roughing/finishing
`N50 T1 G17 S4000 *`	Call toll: drill
`N60 G00 G40 G90 Z+250 *`	Retract the tool
`N70 G37 P01 1 P02 2 P03 3 P04 4 *`	Define contour subprogram
`N80 G120 CONTOUR DATA`	Define general machining parameters
`Q1=-20 ;MILLING DEPTH`	
`Q2=1 ;TOOL PATH OVERLAP`	
`Q3=+0 ;ALLOWANCE FOR SIDE`	
`Q4=+0 ;ALLOWANCE FOR FLOOR`	
`Q5=+0 ;SURFACE COORDINATE`	
`Q6=2 ;SET-UP CLEARANCE`	
`Q7=+100 ;CLEARANCE HEIGTH`	
`Q8=0.1 ;ROUNDING RADIUS`	
`Q9=-1 ;DIRECTION OF ROTATION`	

N90 G121 PILOT DRILLING	Cycle definition: Pilot drilling
Q10=5 ;PLUNGING DEPTH	
Q11=250 ;FEED RATE FOR PLUNGING	
Q13=0 ;ROUGH-OUT TOOL	
N100 G79 M3 *	Cycle call: Pilot drilling
N110 Z+250 M6 *	Tool change
N120 T2 G17 S3000 *	Call the tool for roughing/finishing
N130 G122 ROUGH-OUT	Cycle definition: Coarse roughing
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLUNGING	
Q12=350 ;FEED RATE FOR ROUGHING	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
N140 G79 M3 *	Cycle call: Rough-out
N150 G123 FLOOR FINISHING	Cycle definition: Floor finishing
Q11=100 ;FEED RATE FOR PLUNGING	
Q12=200 ;FEED RATE FOR ROUGHING	
N160 G79 *	Cycle call: Floor finishing
N170 G124 SIDE FINISHING	Cycle definition: Side finishing
Q9=+1 ;DIRECTION OF ROTATION	
Q10=-5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLUNGING	
Q12=400 ;FEED RATE FOR ROUGHING	
Q14=0 ;ALLOWANCE FOR SIDE	
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



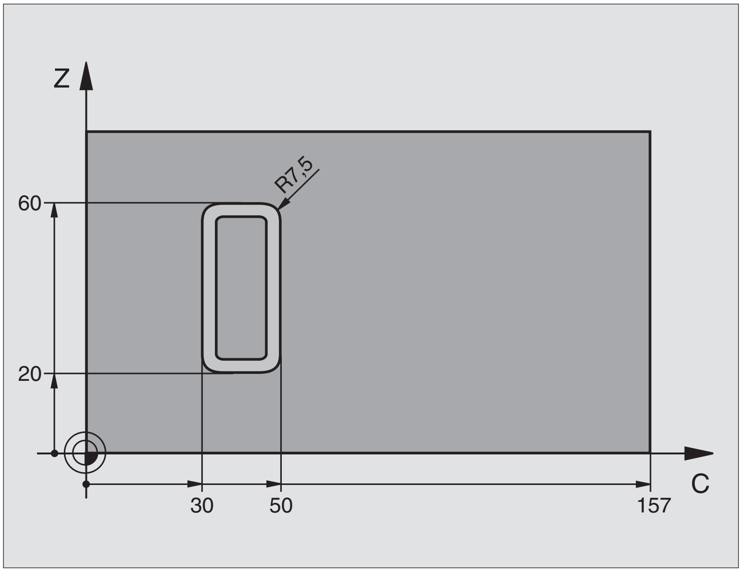
%C25 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define the tool
N50 T1 G17 S2000 *	Tool call
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 *	Define contour subprogram
N80 G125 CONTOUR TRAIN	Define machining parameters
Q1=-20 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q5=+0 ;SURFACE COORDINATE	
Q7=+250 ;CLEARANCE HEIGTH	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLUNGING	
Q12=200 ;FEED RATE FOR MILLING	
Q15=+1 ;CLIMB OR UP-CUT	
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 with Cycle G127

Note:

- Cylinder centered on rotary table
- Datum at center of rotary table



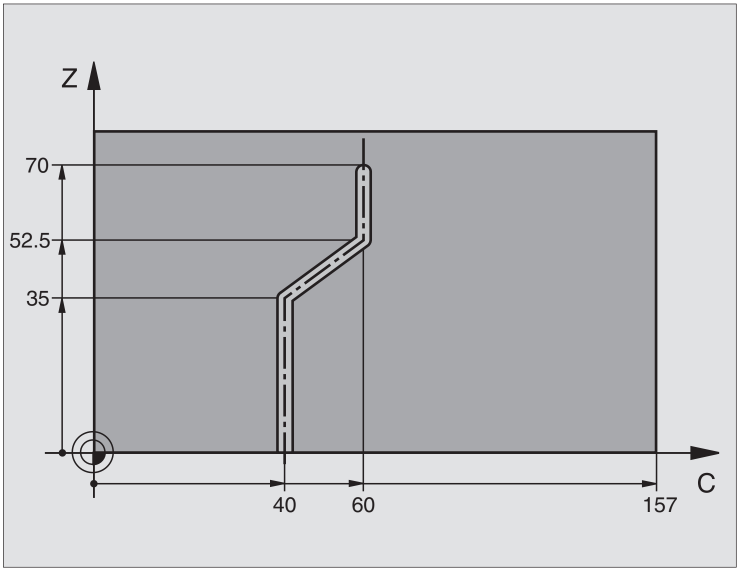
`%C27 G71 *`	
`N10 G99 T1 L+0 R+3.5 *`	Define the tool
`N20 T1 G18 S2000 *`	Call tool, tool axis is Y
`N30 G00 G40 G90 Y+250 *`	Retract the tool
`N40 G37 P01 1 *`	Define contour subprogram
`N70 G127 CYLINDER SURFACE`	Define machining parameters
`    Q1=-7          ;MILLING DEPTH`	
`    Q3=+0          ;ALLOWANCE FOR SIDE`	
`    Q6=2           ;SET-UP CLEARANCE`	
`    Q10=4          ;PLUNGING DEPTH`	
`    Q11=100        ;FEED RATE FOR PLUNGING`	
`    Q12=250        ;FEED RATE FOR MILLING`	
`    Q16=25         ;RADIUS`	
`    Q17=1          ;DIMENSION TYPE`	
`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 entered in degrees
N110 C+114.65 Z+20 *	Drawing dimensions are converted from mm to degrees (157 mm = 360°)
N120 G25 R7.5 *	
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 *	

Example: Cylinder surface with Cycle G128

Notes:

- Cylinder centered on rotary table
- Datum at center of rotary table
- Description of the midpoint path in the contour subprogram



`%C28 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 X+0 *`	Position tool on rotary table center
`N60 G128 CYLINDER SURFACE`	Define machining parameters
`Q1=-7 ;MILLING DEPTH`	
`Q3=+0 ;ALLOWANCE FOR SIDE`	
`Q6=2 ;SET-UP CLEARANCE`	
`Q10=-4 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=250 ;FEED RATE FOR MILLING`	
`Q16=25 ;RADIUS`	
`Q17=1 ;DIMENSION TYPE`	
`Q20=10 ;SLOT WIDTH`	
`N70 C+0 M3 *`	Pre-position rotary table
`N80 G79 *`	Call the cycle
`N90 G00 G40 Y+250 M2 *`	Retract in the tool axis, end program

`N100 G98 L1 *`	Contour subprogram, description of the midpoint path
`N100 G01 G41 C+40 Z+0 *`	Data for the rotary axis are entered in mm (Q17=1)
`N110 Z+35 *`	
`N120 C+60 Z+52.5 *`	
`N130 Z+70 *`	
`N140 G98 L0 *`	
`N999999 %C28 G71 *`	

8.8 SL Cycles with Contour Formula

Fundamentals

SL Cycles and the contour formula enable you to form complex contours by combining subcontours (pockets or islands). You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the complete contour from the selected subcontours, which you link together through a contour formula.

> The memory capacity for programming an SL cycle (all contour description programs) is limited to 32 contours. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. For example, you can program up to approx. 1024 line blocks.
>
> The SL Cycles with contour formula presuppose a structured program layout and enable you to save frequently used contours in individual programs. Using the contour formula, you can connect the subcontours to a complete contour and define whether it applies to a pocket or island.
>
> In its present form, the "SL Cycles with contour formula" function requires input from several areas in the TNC's user interface. This function is to serve as a basis for further development.

Properties of the subcontours

- By default, the TNC assumes that the contour is a pocket. Do not program a radius compensation. In the contour formula you can convert a pocket to an island by making it negative.
- The TNC ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- 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 the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.

Example: Program structure: Machining with SL Cycles and contour formula

```
%CONTOUR G71
...
N50 %:CNT: "MODEL"
N60 G120 Q1= ...
N70 G122 Q10= ...
N80 G79
...
N120 G123 Q11= ...
N130 G79
...
N160 G124 Q9= ...
N170 G79
N180 G00 G40 G90 Z+250 M2
N99999999 %CONTOUR G71
```

Example: Program structure: Calculation of the subcontours with contour formula

```
%MODEL G71
N10 DECLARE CONTOUR QC1 = "ARC1"
N20 DECLARE CONTOUR QC2 = "ARC31XY"
N30 DECLARE CONTOUR QC3 = "TRIANGLE"
N40 DECLARE CONTOUR QC4 = "SQUARE"
N50 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
N99999999 %MODEL G71

%ARC1 G71
N10 I+75 J+50
N20 G11 R+45 H+0 G40
N30 G13 G91 H+360
N99999999 %ARC1 G71

%ARC31XY G71
...
...
```

- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

> 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 G120.

Selecting a program with contour definitions

With the **%:CNT** function you select a program with contour definitions, from which the TNC takes the contour descriptions:

PGM CALL

- To select the functions for program call, press the PGM CALL key.

SELECT CONTOUR

- Press the SELECT CONTOUR soft key.
- Enter the full name of the program with the contour definition and confirm with the END key.

> Program a %:CNT block before the SL Cycles. Cycle 14 CONTOUR GEOMETRY is no longer necessary if you use %:CNT.

Defining contour descriptions

With the **DECLARE CONTOUR** function you enter in a program the path for programs from which the TNC draws the contour descriptions:

DECLARE

- Press the DECLARE soft key.

CONTOUR

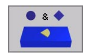
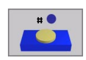
- Press the CONTOUR soft key.
- Enter the number for the contour designator **QC,** and confirm with the ENT key.
- Enter the full name of the program with the contour description and confirm with the END key.

> With the given contour designators QC you can include the various contours in the contour formula.
>
> With the **DECLARE STRING** function you define a text. For the time being, this function is not evaluated.

Entering a contour formula

You can use soft keys to interlink various contours in a mathematical formula.

- Select a Q parameter function: Press the Q key (in the numerical keypad at right). The Q parameter functions are displayed in a soft-key row.
- To select the function for entering the contour formula, press the CONTOUR FORMULA soft key. The TNC then shows the following soft keys:

Logic command	Soft key
Intersected with e.g. `QC10 = QC1 & QC5`	
Joined with e.g. `QC25 = QC7 | QC18`	
Joined without intersection e.g. `QC12 = QC5 ^ QC25`	
Joined with complement of e.g. `QC25 = QC1 \ QC2`	
Complement of contour area e.g. `Q12 = #Q11`	
Opening parenthesis e.g. `QC12 = QC1 * (QC2 + QC3)`	
Closing parenthesis e.g. `QC12 = QC1 * (QC2 + QC3)`	

Overlapping contours

By default, the TNC considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

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 description programs that are defined in a contour definition program. The contour definition program is called through the `%:CNT` function in the actual main program.

Pockets A and B overlap.

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

The pockets are programmed as full circles.

Contour description program 1: Pocket A

```
%POCKET_A G71
N10 G01 X+10 Y+50 G40
N20 I+35 J+50
N30 G02 X+10 Y+50
N999999999 %POCKET_A G71
```

Contour description program 2: Pocket B

```
%POCKET_B G71
N10 G01 X+90 Y+50 G40
N20 I+65 J+50
N30 G02 X+90 Y+50
N999999999 %POCKET_B G71
```

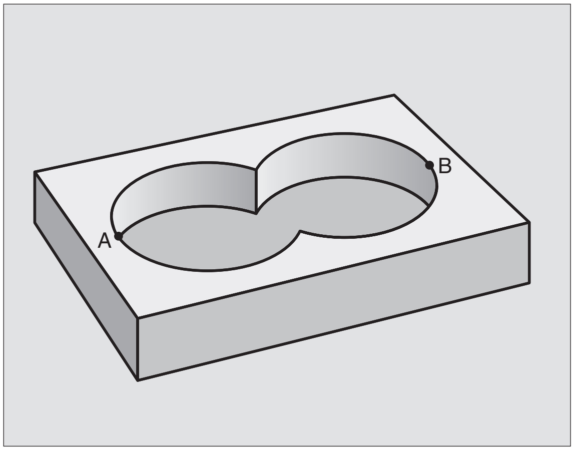
Area of inclusion

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

- The surfaces A and B must be programmed in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the "joined with" function.

Contour definition program:

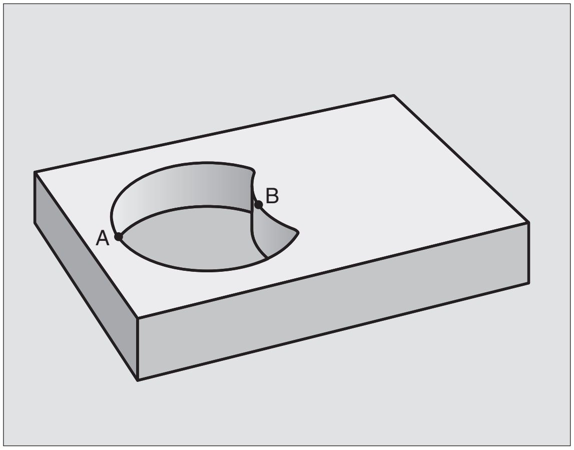
```
N50 ...
N60 ...
N70 DECLARE CONTOUR QC1 = "POCKET_A.H"
N80 DECLARE CONTOUR QC2 = "POCKET_B.H"
N90 QC10 = QC1 | QC2
N100 ...
N110 ...
```



Area of exclusion

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

- The surfaces A and B must be programmed in separate programs without radius compensation.
- In the contour formula, the surface B is subtracted from the surface A with the "joined with complement of" function.



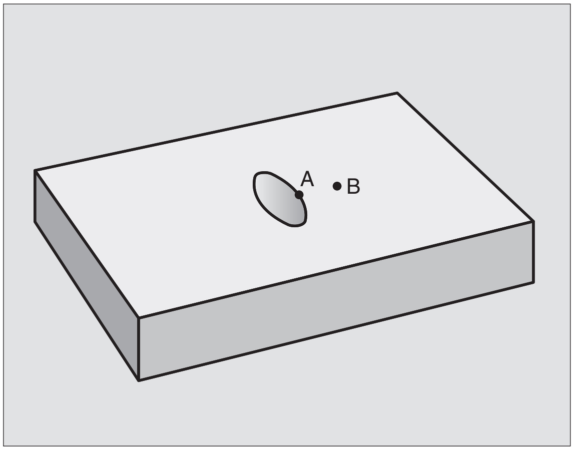
Contour definition program:

```
N50 ...
N60 ...
N70 DECLARE CONTOUR QC1 = "POCKET_A.H"
N80 DECLARE CONTOUR QC2 = "POCKET_B.H"
N90 QC10 = QC1 \ QC2
N100 ...
N110 ...
```

Area of intersection

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

- The surfaces A and B must be programmed in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the "intersection with" function.



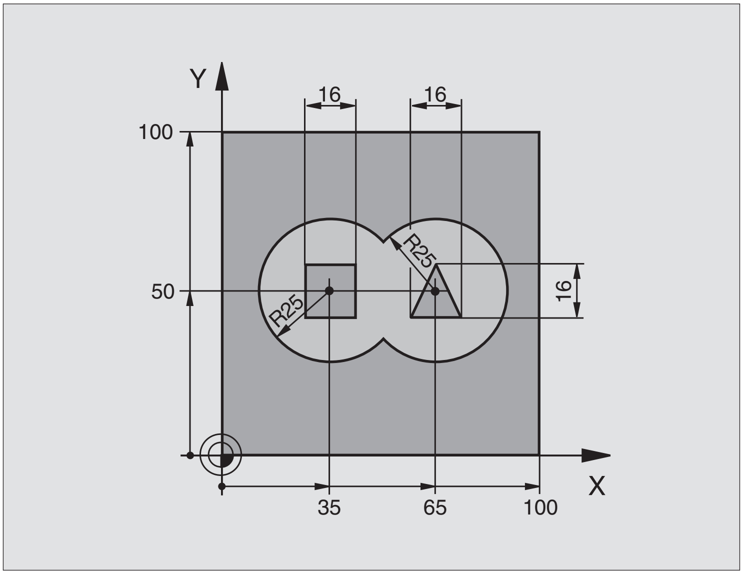
Contour definition program:

```
N50 ...
N60 ...
N70 DECLARE CONTOUR QC1 = "POCKET_A.H"
N80 DECLARE CONTOUR QC2 = "POCKET_B.H"
N90 QC10 = QC1 & QC2
N100 ...
N110 ...
```

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles G120 to G124 (see "SL Cycles Group II" on page 306).

Example: Roughing and finishing superimposed contours with the contour formula



%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+2.5 *	Tool definition of roughing cutter
N40 G99 T2 L+0 R+3 *	Tool definition of finishing cutter
N50 T1 G17 S2500 *	Tool call of roughing cutter
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 %:CNT: "MODEL" *	Specify contour definition program
N80 G120 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGTH	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION OF ROTATION	

`N90 G122 ROUGH-OUT`	Cycle definition: Rough-out
`Q10=5 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=350 ;FEED RATE FOR ROUGHING`	
`Q18=0 ;COARSE ROUGHING TOOL`	
`Q19=150 ;RECIPROCATION FEED RATE`	
`N100 G79 M3 *`	Cycle call: Rough-out
`N110 T2 G17 S5000 *`	Tool call of finishing cutter
`N150 G123 FLOOR FINISHING`	Cycle definition: Floor finishing
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=200 ;FEED RATE FOR ROUGHING`	
`N160 G79 *`	Cycle call: Floor finishing
`N170 G124 SIDE FINISHING`	Cycle definition: Side finishing
`Q9=+1 ;DIRECTION OF ROTATION`	
`Q10=-5 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=400 ;FEED RATE FOR ROUGHING`	
`Q14=0 ;ALLOWANCE FOR SIDE`	
`N180 G79 *`	Cycle call: Side finishing
`N190 G00 Z+250 M2 *`	Retract in the tool axis, end program
`N999999 %C21 G71 *`	

Contour definition program with contour formula:

`%MODEL G71 *`	Contour definition program
`N10 DECLARE CONTOUR QC1 = "ARC1" *`	Definition of the contour designator for the program “CIRCLE1“
`N20 D00 Q1 P01 +35 *`	Assignment of values for parameters used in PGM “CIRCLE31XY“
`N30 D00 Q2 P01 +50 *`	
`N40 D00 Q3 P01 +25 *`	
`N50 DECLARE CONTOUR QC2 = "ARC31XY" *`	Definition of the contour designator for the program “CIRCLE31XY”
`N60 DECLARE CONTOUR QC3 = "TRIANGLE" *`	Definition of the contour designator for the program “TRIANGLE”
`N70 DECLARE CONTOUR QC4 = "SQUARE" *`	Definition of the contour designator for the program “SQUARE”
`N80 QC10 = ( QC 1 | QC 2 ) \ QC 3 \ QC 4 *`	Contour formula
`N99999999 %MODEL G71 *`	

Contour description programs:

`%ARC1 G71 *`	Contour description program: circle at right
`N10  I+65 J+50 *`	
`N20  G11 R+25 H+0 G40 *`	
`N30  CP IPA+360 DR+ *`	
`N99999999 %ARC1 G71 *`	

`%ARC31XY G71 *`	Contour description program: circle at left
`N10  I+Q1 J+Q2 *`	
`N20  G11 R+Q3 H+0 G40 *`	
`N30  G13 G91H+360 *`	
`N99999999 %ARC31XY G71 *`	

`%TRIANGLE G71 *`	Contour description program: triangle at right
`N10  G01 X+73 Y+42 G40 *`	
`N20  G01 X+65 Y+58 *`	
`N30  G01 X+42 Y+42 *`	
`N40  G01 X+73 *`	
`N99999999 %TRIANGLE G71 *`	

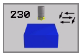
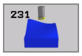
`%SQUARE G71 *`	Contour description program: square at left
`N10 G01 X+27 Y+58 G40 *`	
`N20 G01 X+43 *`	
`N30 G01 Y+42 *`	
`N40 G01 X+27 *`	
`N50 G01 Y+58 *`	
`N99999999%SQUARE G71 *`	

8.9 Cycles for Multipass Milling

Overview

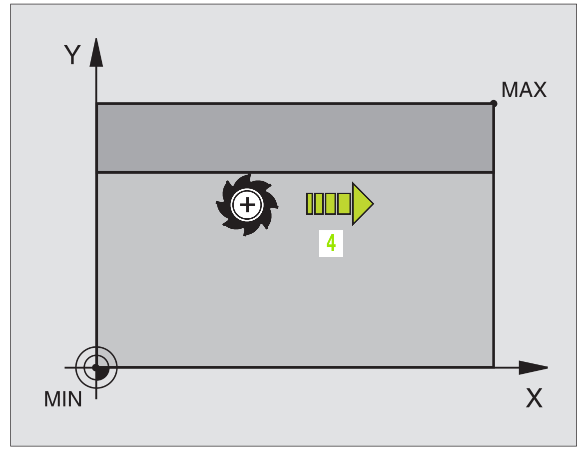
The TNC offers three cycles for machining the following surface types:

- Created 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 3-D DATA For multipass milling of 3-D data in several infeeds	
G230 MULTIPASS MILLING For flat rectangular surfaces	
G231 RULED SURFACE For oblique, inclined or twisted surfaces	

RUN 3-D DATA (Cycle G60)

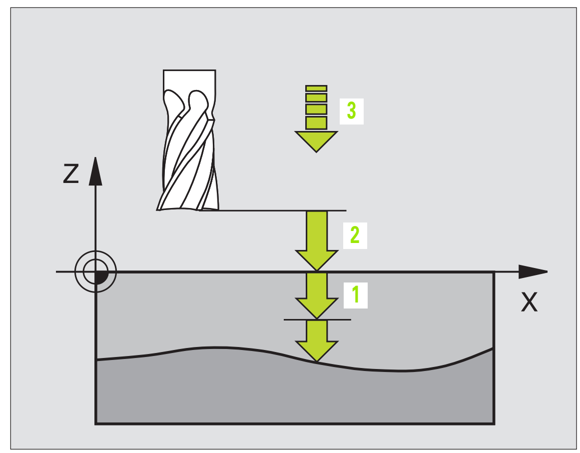
1. From the current position, the TNC positions the tool in rapid traverse in the tool axis to the set-up clearance above the MAX point that you have programmed in the cycle.
2. The tool then moves in rapid traverse in the working plane to the MIN point you have programmed in the cycle.
3. From this 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 3-D 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:

Cycle G60 allows you to run 3-D data using several infeeds which have been created with an off-line programming system.

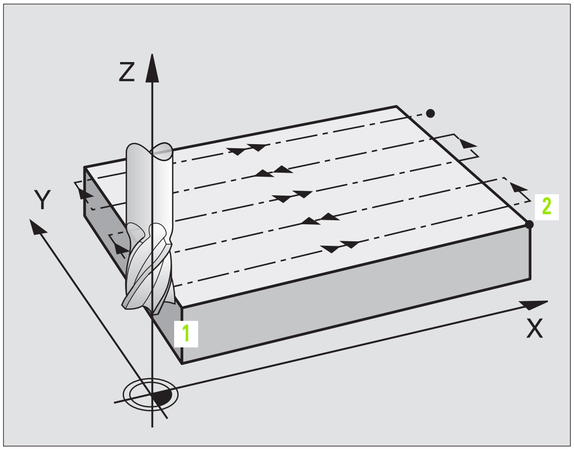
- **`PGM Name 3-D data`**: Enter the name of the file in which the data is stored. If the file is not stored in the current directory, enter the complete path.
- **`Min. point of range:`** Lowest coordinates (X, Y and Z coordinates) in the range to be milled.
- **`Max. point of range:`** Largest coordinates (X, Y and Z coordinates) in the range to be milled.
- **`Set-up clearance`** 1 (incremental value): Distance between tool tip and workpiece surface for tool movements in rapid traverse.
- **`Plunging depth`** 2 (incremental value): Infeed per cut.
- **`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 P01 X+0 P02 Y+0
    P03 Z-20 P04 X+100 P05 Y+100 P06 Z+0
    P07 2 P08 +5 P09 100 P10 350 M13 *
```

MULTIPLASS MILLING (Cycle G230)



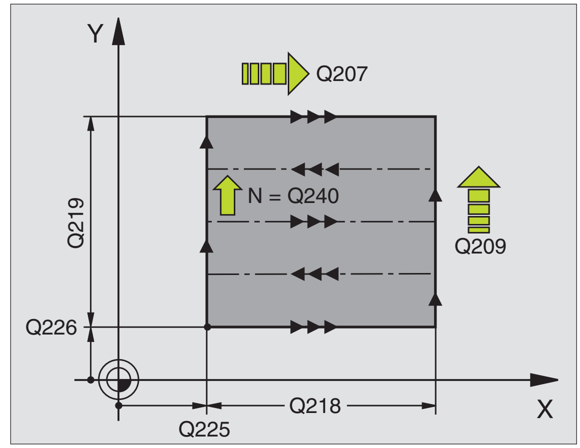
1. From the current position in the working plane, the TNC positions the tool at rapid traverse to the starting point **1**; the TNC moves the tool by its radius to the left and upward.
2. The tool then moves in rapid traverse in the tool axis to set-up clearance. From there it approaches the programmed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

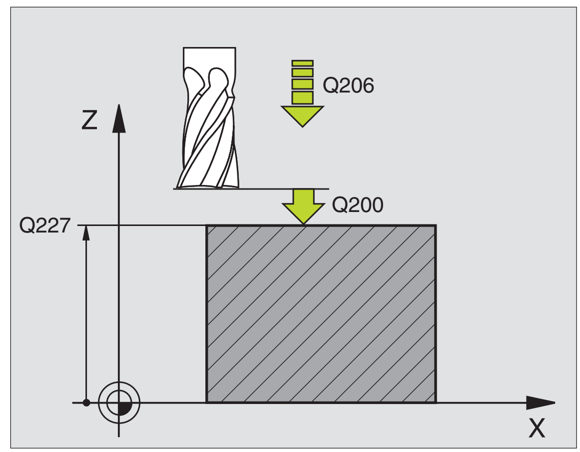
Before programming, note the following:

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

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

- **`Starting point in 1st axis`** Q225 (absolute value): Minimum point coordinate of the surface to be multipass-milled in the reference axis of the working plane.
- **`Starting point in 2nd axis`** Q226 (absolute value): Minimum-point coordinate of the surface to be multipass-milled in the minor axis of the working plane.
- **`Starting point in 3rd axis`** Q227 (absolute value): Height in the spindle axis at which multipass-milling is carried out.
- **`First side length`** Q218 (incremental value): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in the 1st axis.
- **`Second side length`** Q219 (incremental value): Length of the surface to be multipass-milled in the minor axis of the working plane, referenced to the starting point in the 2nd axis.
- **`Number of cuts`** Q240: Number of passes to be made over the width.
- **`Feed rate for plunging`** 206: Traversing speed of the tool in mm/min while penetrating from the 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.



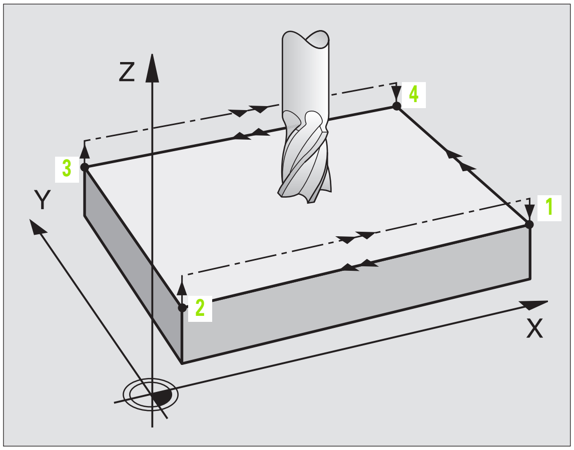


Example: NC block

```
N71 G230 MULTIPASS MILLING
    Q225=+10      ;STARTING PNT 1ST AXIS
    Q226=+12      ;STARTING PNT 2ND AXIS
    Q227=+2.5     ;STARTING PNT 3RD AXIS
    Q218=150      ;FIRST SIDE LENGTH
    Q219=75       ;SECOND SIDE LENGTH
    Q240=25       ;NUMBER OF CUTS
    Q206=150      ;FEED RATE FOR PLUNGING
    Q207=500      ;FEED RATE FOR MILLING
    Q209=200      ;STEPOVER FEED RATE
    Q200=2        ;SET-UP CLEARANCE
```

RULED SURFACE (Cycle G231)

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

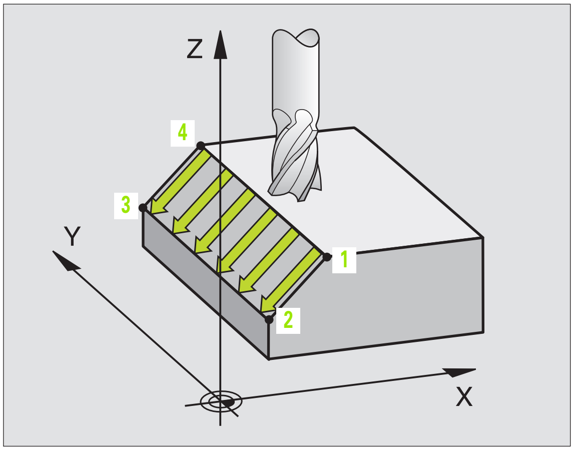


Cutting motion

The starting point, and therefore the milling direction, is selectable because the TNC always moves from point **1** to point **2** and in the total movement from point **1** / **2** to point **3** / **4**. You can program point **1** at any corner of the surface to be machined.

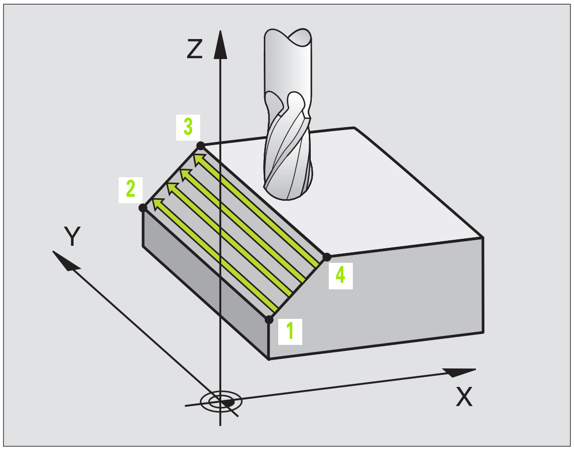
If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

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



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

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

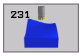


Before programming, note the following:

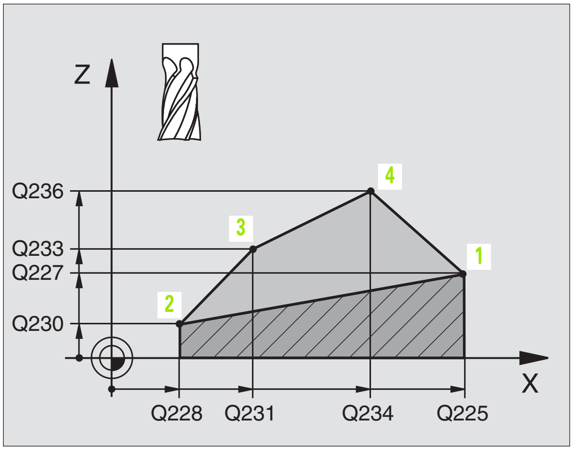
From the current position, the TNC positions the tool in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

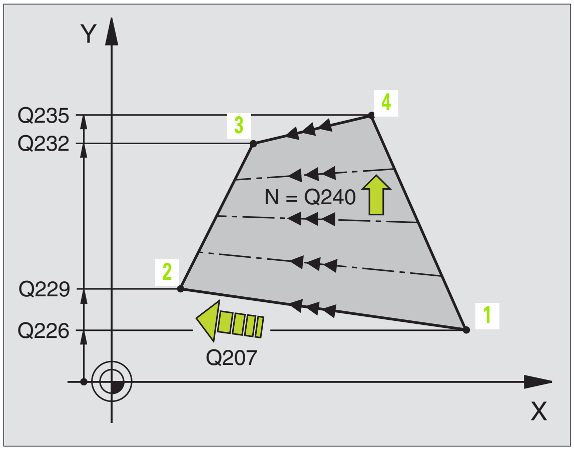
The TNC moves the tool with radius compensation **G40** to the programmed positions.

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



- **Starting point in 1st axis** Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the reference axis of the working plane.
- **Starting point in 2nd axis** Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane.
- **Starting point in 3rd axis** Q227 (absolute value): Starting point coordinate of the surface to be multipass-milled in the tool axis.
- **2nd point in 1st axis** Q228 (absolute value): Stopping point coordinate of the surface to be multipass milled in the reference axis of the working plane.
- **2nd point in 2nd axis** Q229 (absolute value): Stopping point coordinate of the surface to be multipass milled in the minor axis of the working plane.
- **2nd point in 3rd axis** Q230 (absolute value): Stopping point coordinate of the surface to be multipass milled in the tool axis.
- **3rd point in 1st axis** Q231 (absolute value): Coordinate of point **3** in the reference axis of the working plane.
- **3rd point in 2nd axis** Q232 (absolute value): Coordinate of point **3** in the minor axis of the working plane.
- **3rd point in 3rd axis** Q233 (absolute value): Coordinate of point **3** in the tool axis



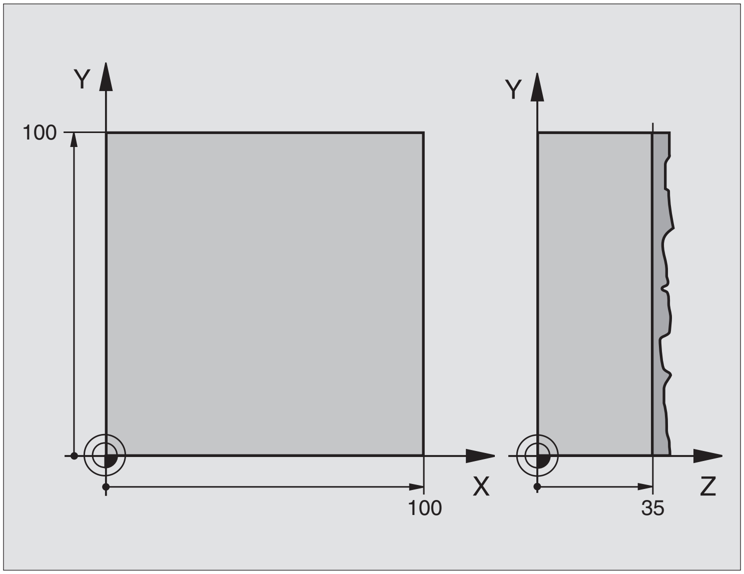


- **4th point in 1st axis** Q234 (absolute value): Coordinate of point **4** in the reference axis of the working plane.
- **4th point in 2nd axis** Q235 (absolute value): Coordinate of point **4** in the minor axis of the working plane.
- **4th point in 3rd axis** Q236 (absolute value): Coordinate of point **4** in the tool axis.
- **Number of cuts** Q240: Number of passes to be made between points **1** and **4**, **2** and **3**.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. The TNC performs the first step at half the programmed feed rate.

Example: NC blocks

```
N72 G231 RULED SURFACE
    Q225=+0         ;STARTING PNT 1ST AXIS
    Q226=+5         ;STARTING PNT 2ND AXIS
    Q227=-2         ;STARTING PNT 3RD AXIS
    Q228=+100       ;2ND POINT 1ST AXIS
    Q229=+15        ;2ND POINT 2ND AXIS
    Q230=+5         ;2ND POINT 3RD AXIS
    Q231=+15        ;3RD POINT 1ST AXIS
    Q232=+125       ;3RD POINT 2ND AXIS
    Q233=+25        ;3RD POINT 3RD AXIS
    Q234=+15        ;4TH POINT 1ST AXIS
    Q235=+125       ;4TH POINT 2ND AXIS
    Q236=+25        ;4TH POINT 3RD AXIS
    Q240=40         ;NUMBER OF CUTS
    Q207=500        ;FEED RATE FOR MILLING
```

Example: Multipass milling



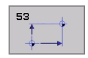
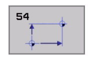
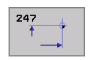
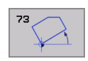
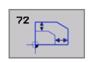
%C230 G71	
N10 G30 G17 X+0 Y+0 Z+0 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+40 *	
N30 G99 T1 L+0 R+5 *	Define the tool
N40 T1 G17 S3500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G230 MULTIPASS MILLING	Cycle definition: MULTIPASS MILLING
N60 G230 MULTIPASS MILLING	Cycle definition: MULTIPASS MILLING
Q225=+0 ;STARTING PNT 1ST AXIS	
Q226=+0 ;STARTING PNT 2ND AXIS	
Q227=+35 ;STARTING PNT 3RD AXIS	
Q218=100 ;FIRST SIDE LENGTH	
Q219=100 ;SECOND SIDE LENGTH	
Q240=25 ;NUMBER OF CUTS	
Q206=250 ;FEED RATE FOR PLUNGING	
Q207=400 ;FEED RATE FOR MILLING	
Q209=150 ;STEPOVER FEED RATE	
Q200=2 ;SET-UP CLEARANCE	

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 *	

8.10 Coordinate Transformation Cycles

Overview

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

Cycle	Soft key
G53/G54 DATUM SHIFT For shifting contours directly within the program or from datum tables	 
G247 DATUM SETTING Datum setting during program run	
G28 MIRROR IMAGE 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 for machines with tilting heads and/or rotary tables	

Effect of coordinate transformations

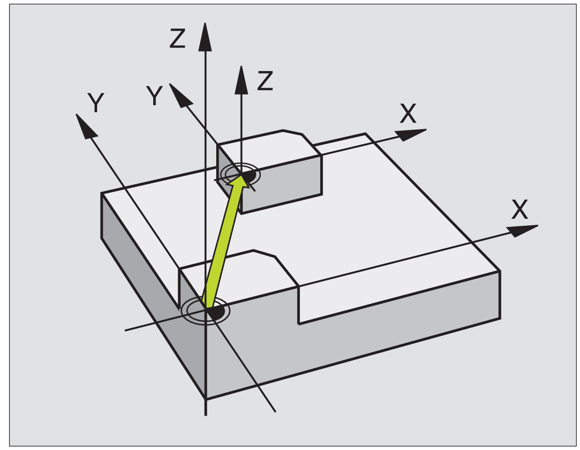
Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0.
- Execute a miscellaneous function M02, M30, or an N999999 %... block (depending on MP7300).
- Select a new program.
- Program miscellaneous function M142 Erasing modal program information.

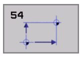
DATUM SHIFT (Cycle G54)

A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.



Effect

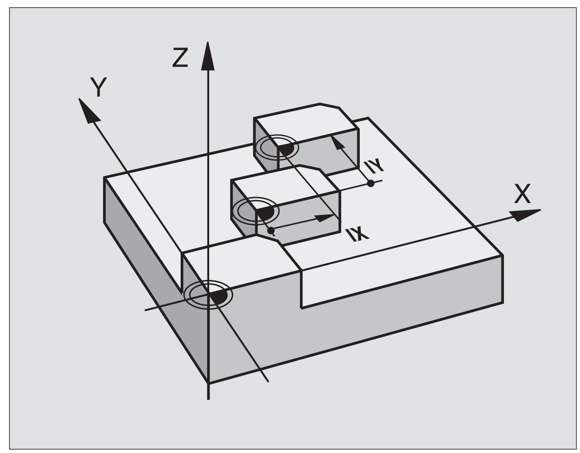
When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.



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

Cancellation

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



Graphics

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.
- All of the 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 Z+100 *
...
N78 G54 G90 REF X+25 Y-12.5 Z+100 *
```

DATUM SHIFT with datum tables (Cycle G53)

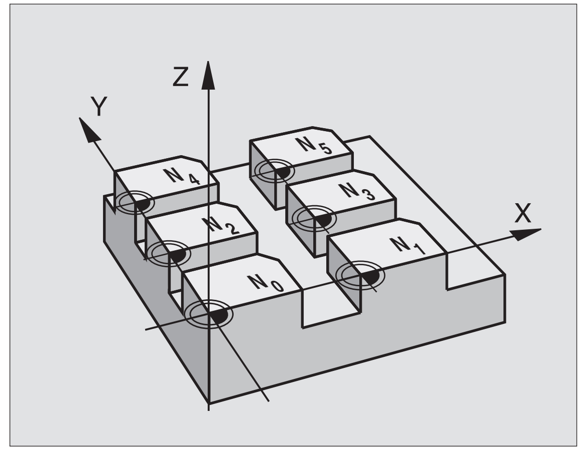
If you are using datum shifts with datum tables, then use the Select Table function to activate the desired datum table from the NC program.

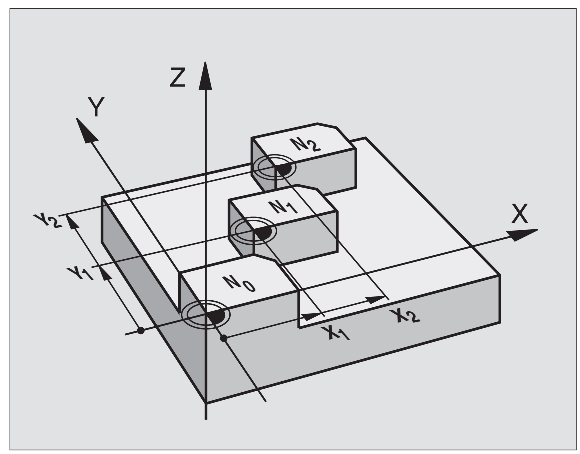
If you work without the Select Table block **%:TAB:**, you must activate the desired datum table before the test run or the program run (This applies also for the programming graphics.):

- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table receives the status S.
- Use the file management in a program run mode to select the desired table for a program run: The table receives the status M.

The coordinate values from datum tables are only effective with absolute coordinate values.

New lines can only be inserted at the end of the table.



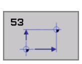


Function

Datum tables are used for

- frequently recurring machining sequences at various locations on the workpiece
- frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.



▶ **Shift: Table row?** P01: 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.

Cancellation

- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.
- Execute a datum shift to the coordinates X=0; Y=0 etc. directly with a cycle definition.

Example: NC blocks

```
N72 G53 P01 12 *
```

Selecting a datum table in the part program

With the Select Table (**%:TAB:**) function, you select the datum table from which the TNC takes the datums:

PGM CALL

- To select the functions for program call, press the PGM CALL key.

DATUM TABLE

- Press the TOOL TABLE soft key.
- Enter the complete path name of the datum table and confirm your entry with the END key.

> Program a **%:TAB:** block before Cycle **G53** Datum Shift.
>
> A datum table selected with Select Table remains active until you select another datum table with **%:TAB:** or through PGM MGT.

Editing a datum table

Select the datum table in the **Programming and Editing** mode of operation.

PGM MGT

- To call the file manager, press the PGM MGT key (see “File Management: Fundamentals,” page 69).
- Display the datum tables: Press the soft keys SELECT TYPE and SHOW .D.
- Select the desired table or enter a new file name.
- Edit the file. The soft-key row comprises the following functions for editing:

Function	Soft key
Select beginning of table	BEGIN
Select end of table	END
Go to previous page	PAGE
Go to next page	PAGE
Insert line (only possible at end of table)	INSERT LINE
Delete line	DELETE LINE
Confirm entered line and go to the beginning of next line	NEXT LINE
Add the entered number of lines (reference points) to the end of the table	APPEND N LINES
Select list view (standard) or form view.	LIST FORM

Editing a pocket table in a Program Run operating mode

In a program run mode you can select the active datum table. Press the DATUM TABLE soft key. You can then use the same editing functions as in the **Programming and Editing** mode of operation.

Transferring the actual values into the datum table

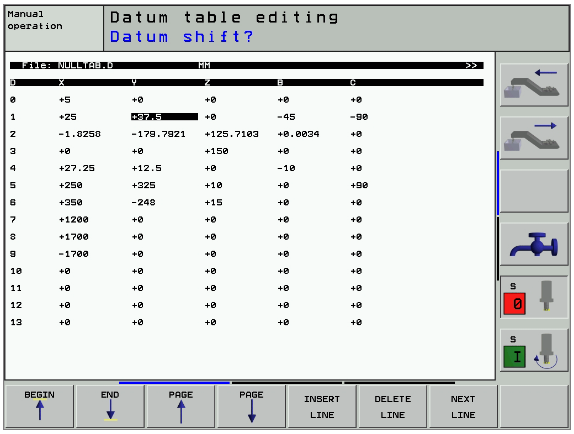
You can enter the current tool position or the last probed position in the datum table by pressing the "actual-position-capture" key:

- Place the text box on the line of the column in which you want to enter the position.
- Select the actual-position-capture function: The TNC opens a pop-up window that asks whether you want to enter the current tool position or the last probed values.
- Select the desired function with the arrow keys and confirm your selection with the ENT key.
- ALL VALUES: To enter the values in all axes, press the ALL VALUES soft key.
- PRESENT VALUE: To enter the value in the axis where the text box is located, press the CURRENT VALUE soft key.

Configuring the datum table

In 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.

If you do not wish to define a datum table for an active axis, press the NO ENT key. The TNC then enters a dash in that column.

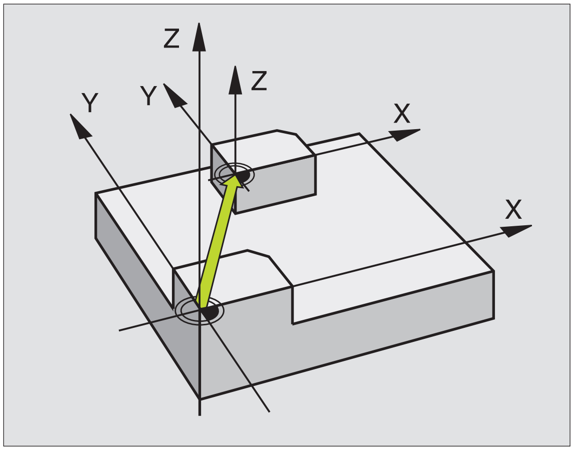


To leave a datum table

Select a different type of file in file management and choose the desired file.

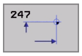
DATUM SETTING (Cycle G247)

With the Cycle DATUM SETTING, you can activate a datum defined in a datum table as the new datum.



Effect

After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new datum. Setting datums for rotary axes is also possible.



- **`Number for datum?`**: Enter the number of the datum in the datum table.

Cancellation

You can reactivate the last datum set in the Manual mode by entering the miscellaneous function M104.

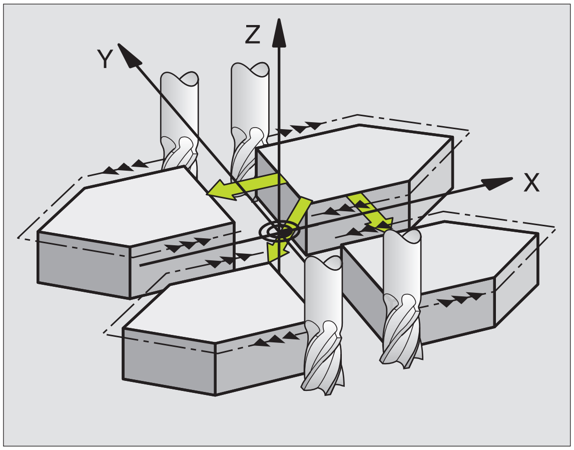
> The TNC only sets the datum for those axes which are active in the datum table. An axis displayed as a column in the datum table, but not existing on the TNC, will cause an error message.
>
> Cycle G247 always interprets the values saved in the datum table as coordinates referenced to the machine datum. Machine parameter 7475 has no influence on this.
>
> When using Cycle G247, you cannot use the block scan function for mid-program startup.
>
> Cycle G247 is not functional in Test Run mode.

Example: NC block

```
N13 G247 DATUM SETTING
     Q339=4          ;DATUM NUMBER
```

MIRROR IMAGE (Cycle G28)

The TNC can machine the mirror image of a contour in the working plane.



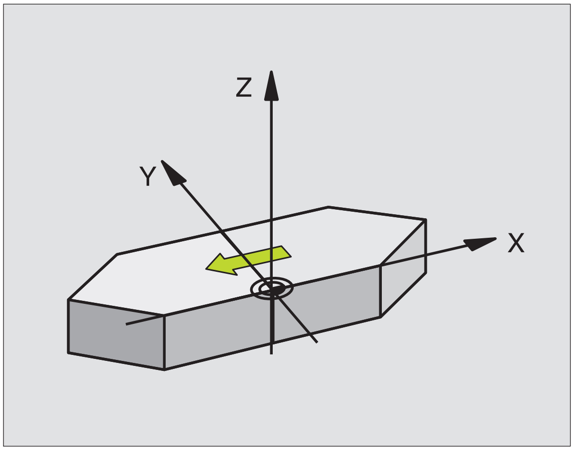
Effect

The mirror image cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

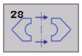
- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

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

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.



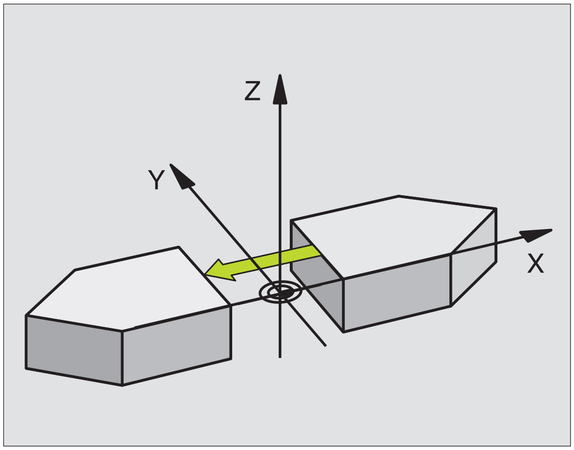
> If you mirror only one axis, the machining direction is reversed for the new machining cycles (cycles 2xx). The machining direction remains the same for older machining cycles, such as Cycle 4 POCKET MILLING.



▶ **Mirrored axis?**: Enter the axes to be mirrored. You can mirror all axes, including rotary axes, except for the spindle axis and its auxiliary axes. You can enter up to three axes.

Reset

Program the MIRROR IMAGE cycle once again with NO ENT.

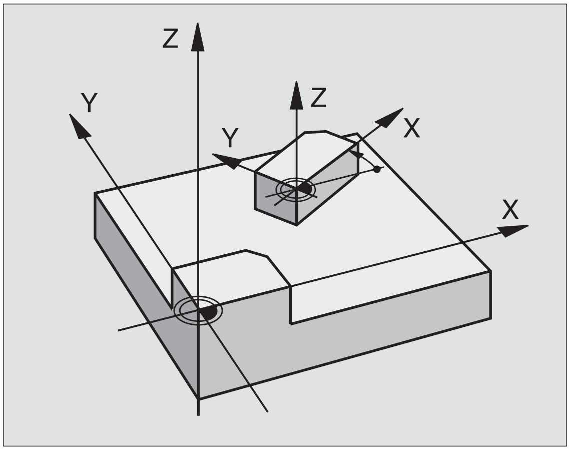


Example: NC block

```
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.

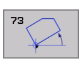
Reference axis for the rotation angle:

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

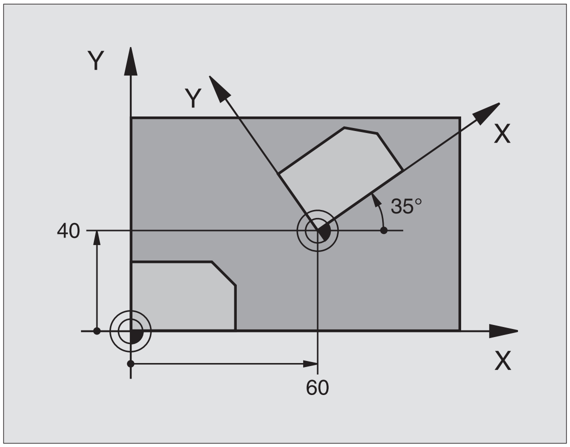
Before programming, note the following:

An active radius compensation is canceled by defining Cycle **G73** and must therefore be reprogrammed, if required.

After defining Cycle **G73,** you must move both axes of the working plane to activate rotation for all axes.



- **Rotation:** Enter the rotation angle in degrees (°). Input range: -360° to +360° (absolute G90 before H or incremental G91 before H).



Cancellation

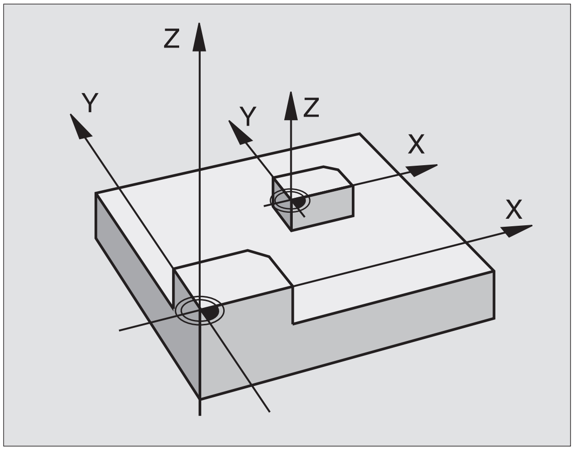
Program the ROTATION cycle once again with a rotation angle of 0°.

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

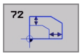
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- the working plane, or on all three coordinate axes at the same time (depending on MP7410)
- the dimensions in cycles
- the parallel axes U,V,W

Prerequisite

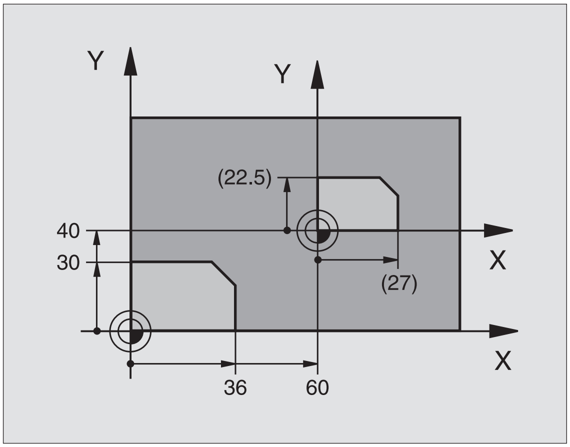
It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.



▶ **`Scaling factor?`**: Enter the scaling factor F. The TNC multiplies the coordinates and radii by the F factor (as described under “Effect” 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 for the same axis.

Example: NC blocks

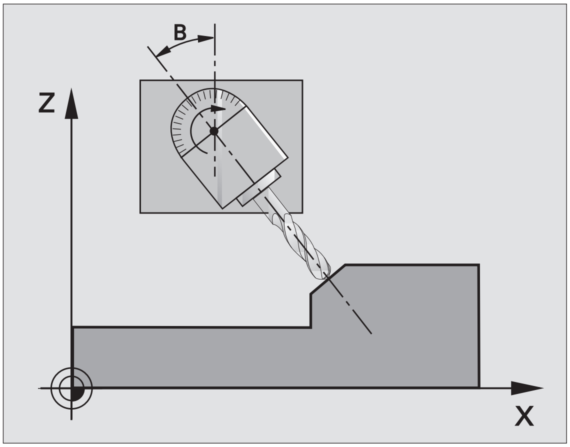
```
N72 G72 F0.750000 *
```

WORKING PLANE (Cycle G80)

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 rotary axes or as mathematical angles of a tilted plane. Refer to your machine manual.

The working plane is always tilted around the active datum.

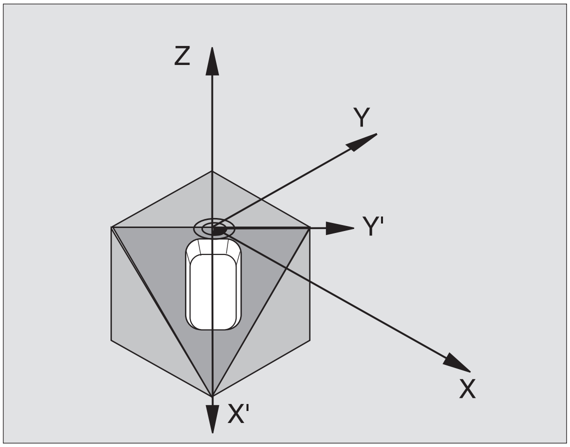
For fundamentals, see "Tilting the Working Plane," page 52. Please read this section completely.



Effect

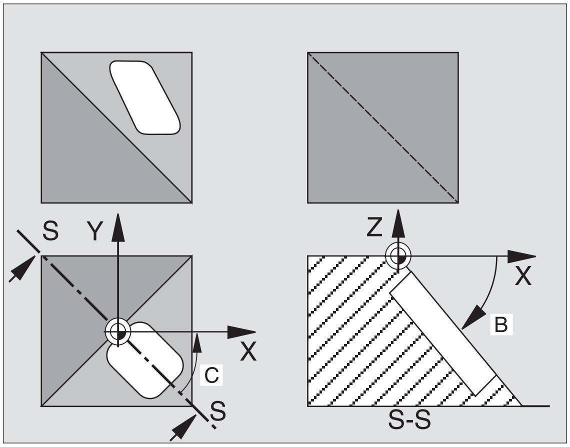
In Cycle **G80** you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the tilting axes directly.
- 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. 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 19 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 must 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 “Tilting the Working Plane,” page 52), the angular value entered in this menu is overwritten by Cycle **G80** WORKING PLANE.

- **Tilt axis and tilt angle?:** Enter 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 value): 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 again, without defining an axis, to disable the function.

Position the 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. Refer to your machine manual.

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.

If the axes are not positioned automatically in Cycle **G80,** position them before defining the cycle, for example with a G01 block.

Example NC blocks:

N50 G00 G40 Z+100 *	
N60 X+25 Y+10 *	
N70 G01 A+15 F1000 *	Position the 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 necessary, the TNC outputs 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 “Miscellaneous Functions for Coordinate Data,” page 188).

Positioning movements with straight lines that are referenced to the machine coordinate system (blocks with M91 or M92) can also be executed in a tilted working plane. Constraints:

- Positioning is without length compensation.
- Positioning is without machine geometry compensation.
- Tool radius compensation is not permitted.

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 them:

1. Activate the datum shift.
2nd: Activate tilting function
3rd: Activate rotation
...
Machining
...
1st: Reset the rotation
2nd: Reset the tilting function
3rd: Reset the datum shift

Automatic workpiece measurement in the tilted system

The TNC measuring cycles enable 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 if 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.
- If required, define Cycle **G80** WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle **G80.** You can define the new angular values directly.
- Reset Cycle **G80** WORKING PLANE. Program 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle **G80,** without defining an axis.
- 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 Manual Data Input (MDI)

Pre-position the rotary 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 Operating 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 rotary axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5 Set the datum

- Manually by touching the workpiece with the tool in the untilted coordinate system (see "Datum Setting (Without a 3-D Touch Probe)," page 50).
- Controlled with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles Manual, chapter 2).
- Automatically by using a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles Manual, chapter 3).

6 Start the part program in the operating mode Program Run, Full Sequence

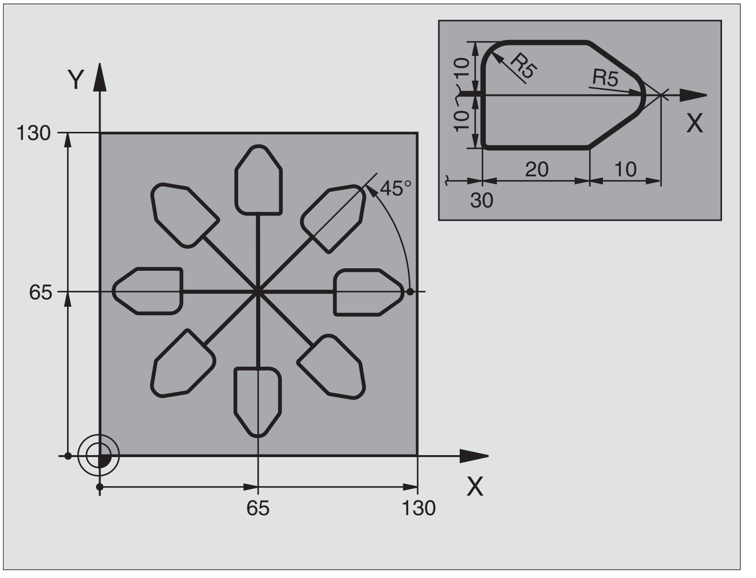
7 Manual Operation mode

Use the 3-D ROT soft key to set the TILT WORKING PLANE function to INACTIVE. Enter an angular value of 0° for each axis in the menu (see "Activating manual tilting," page 55).

Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- For subprograms within a subprogram, see “Subprograms,” page 373.



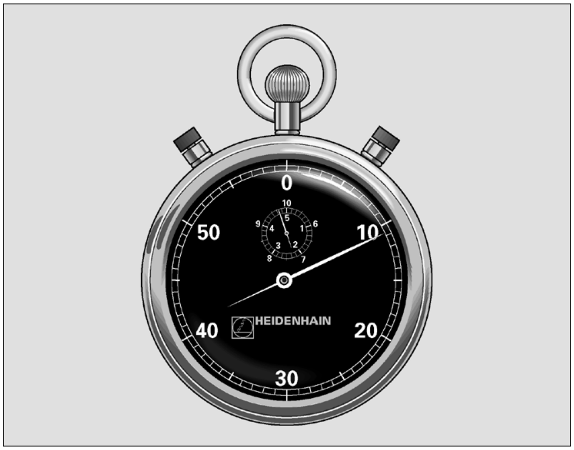
`%KOUMR G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define the workpiece blank
`N20 G31 G90 X+130 Y+130 Z+0 *`	
`N30 G99 T1 L+0 R+1 *`	Define the tool
`N40 T1 G17 S4500 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 G54 X+65 Y+65 *`	Shift datum to center
`N70 L1.0 *`	Call milling operation
`N80 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; repeat 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 *
```

8.11 Special Cycles

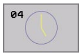
DWELL TIME (Cycle G04)

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



Effect

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



- **`Dwell time in seconds:`** Enter the dwell time in seconds.

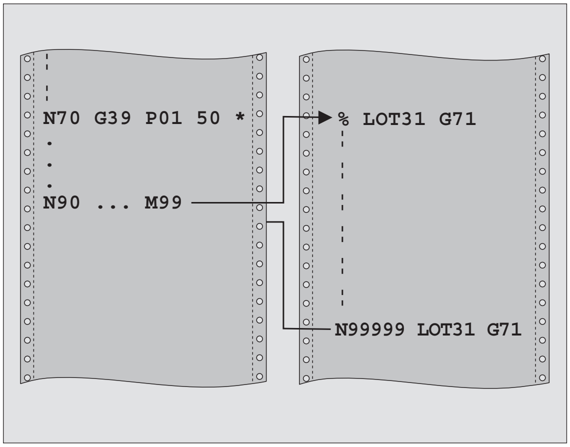
Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

Example: NC block

```
N74 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.

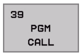


Before programming, note the following:

If you want to define an ISO program to be a cycle, enter the file type .I behind the program name.

If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only to enter the program name.

If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path (for example TNC:\KLAR35\FK1\50.I.



- **`Program name:`** Enter the name of the program you want to call and, if necessary, the directory it is located in.

Call the program with

- **`G79`** (separate block) or
- **`M99`** (blockwise) or
- **`M89`** (executed after every positioning block)

Example: Program call

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

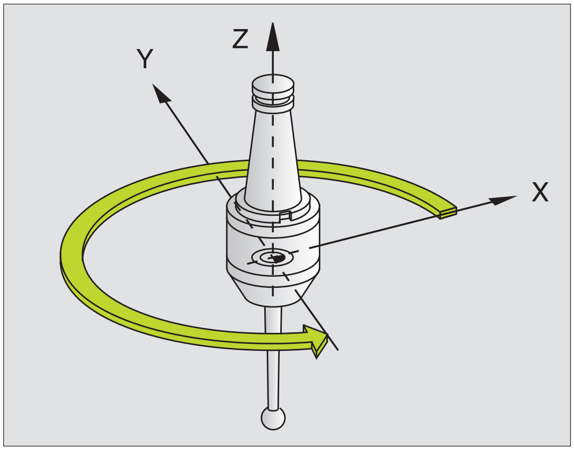
Example: NC blocks

```
N550 G39 P01 50 *
N560 G00 X+20 Y+50 M9 9*
```

ORIENTED SPINDLE STOP (Cycle G36)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Cycle 13 is used internally for machining cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.



The TNC 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

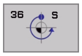
Example: NC block

```
N76 G36 S25*
```

Effect

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 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°

TOLERANCE (Cycle G62)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The input parameters **Finishing/roughing** and **Tolerance for rotary axes** are effective only if the HSC filter (**software option 2**) is active on your machine. The TNC will otherwise display an error message. If necessary, contact your machine tool builder.

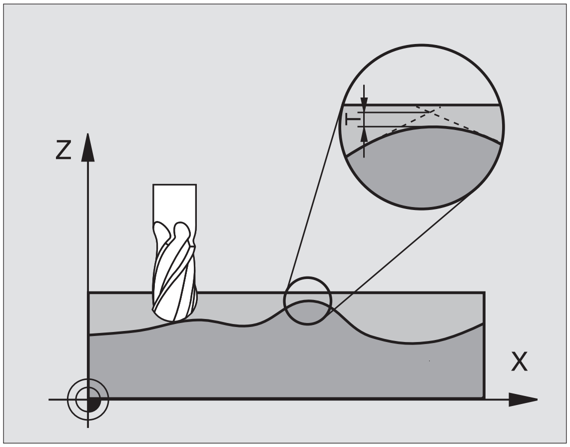
The TNC automatically smoothes the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface. If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the fastest possible speed without short pauses for computing time. As a result the surface quality is improved and the machine is protected.

A contour deviation results from the smoothing. The size of this deviation **(tolerance value)** is set in a machine parameter by the machine manufacturer. With Cycle **G62,** you can change the pre-set tolerance value and select different filter settings.

Before programming, note the following:

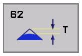
Cycle **G62** is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can reset Cycle **G62** by defining Cycle **G62** again and confirming the dialog question after the **Tolerance value** with NO ENT. Resetting Cycle 32 reactivates the pre-set tolerance:

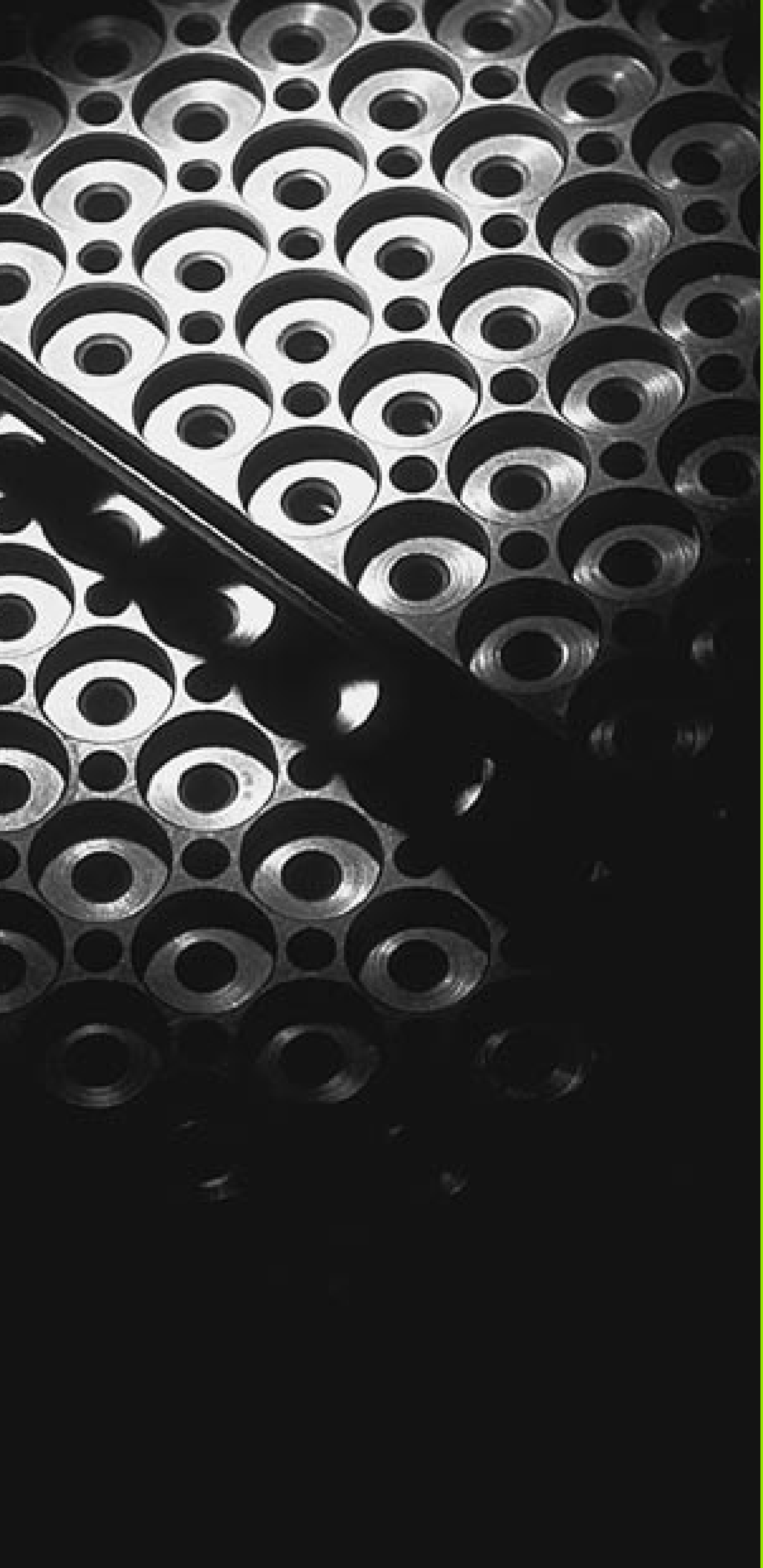


Example: NC block

```
N78 G62 T0.05 P01 0 P02 5*
```



- **`Tolerance for contour deviation:`** Permissible contour deviation in mm (for inch programs in inches)
- **`Finishing=0, Roughing=1:`** Activate filter:
  - Input value 0:
    Milling with increased contour accuracy. The TNC uses the filter settings that your machine tool builder has defined for finishing operations.
  - Input value 1:
    Milling at an increased feed rate. The TNC uses the filter settings that your machine tool builder has defined for roughing operations.
- **`Tolerance for rotary axes:`** Permissible position error of rotary axes in degrees when M128 is active. The TNC always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for programs for more than one axis by entering a large tolerance value (e.g. 10°), since the TNC does not always have to move the rotary axis to the given nominal position. The contour will not be damaged by entering a tolerance value. Only the position of the rotary axis with respect to the workpiece surface will change.



9

Programming: Subprograms and Program Section Repeats

9.1 Labeling Subprograms and Program Section Repeats

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

Labels

Subprograms and program section repeats begin with the function **G98 L** in the part program. The letter L stands for "label."

A label is identified by a number between 1 and 254. Each label number 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.
>
> 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.

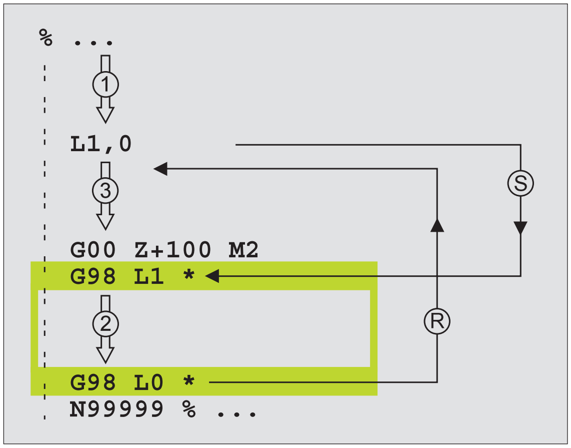
9.2 Subprograms

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 **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

LBL SET

- To mark the beginning, press the LBL SET key.
- Enter the subprogram number and confirm with the END key.
- To mark the end, press the LBL SET key and enter the label number "0".

Calling a subprogram

LBL CALL

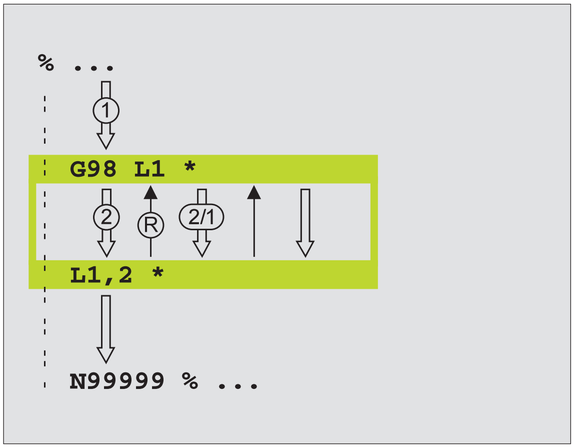
- To call a subprogram, press the LBL CALL key.
- **Label number:** Enter the label number of the subprogram to be called, then confirm with the ENT key.
- **Repeat REP:** Enter "**.0**", then confirm with the ENT key.

L0.0 is not permitted, as it corresponds to the program end call.

9.3 Program Section Repeats

Label G98

The beginning of a program section repeat is marked by the label **G98 L**. 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 **L 1.2** 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 TNC always executes the program section once more than the programmed number of repeats.

Programming a program section repeat

LBL SET

- ▶ To mark the beginning, press the LBL SET key, then confirm with the ENT key.
- ▶ Enter a label number for the program section to be repeated, then confirm with the ENT key.

Calling a program section repeat

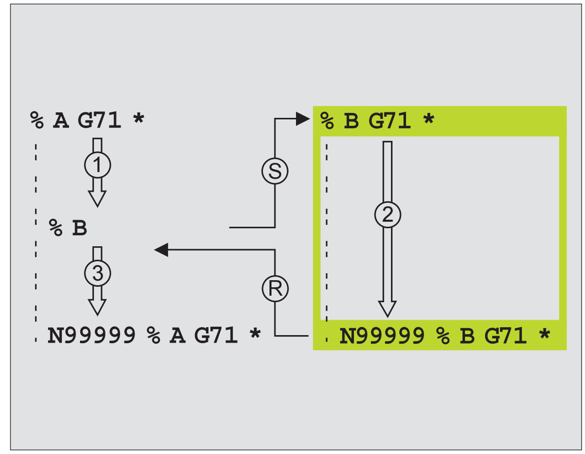
LBL CALL

- ▶ Press the LBL CALL key.
- ▶ **Label number:** Enter the label number of the subprogram to be called, then confirm with the ENT key.
- ▶ **Repeat REP:** Enter the number of repeats, then confirm with the ENT key.

9.4 Separate Program as Subprogram

Operating sequence

1 The TNC executes the part program up to the block in which another program is called with %.
2 Then the other program is run from beginning to end.
3 The TNC then resumes the first (calling) part program with the block after 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 must not contain a call with % into the calling program (endless loop).

Calling any program as a subprogram

PGM CALL

- To select the functions for program call, press the PGM CALL key.

PROGRAM

- Press the PROGRAM soft key.
- Enter the complete path name of the program you want to call and confirm your entry with the END key.

You 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.

The program you are calling must be stored on the hard disk of your TNC.

You need only enter the program name if the program you want to call is located in the same directory as the program you are calling it from.

If the called program is not located in the same directory as the program you are calling it from, you must enter the complete path, e.g. TNC:\ZW35\ROUGH\PGM1.H

9.5 Nesting

Types of nesting

- Subprograms within a subprogram
- Program section repeats within a program section repeat
- Subprograms repeated
- Program section repeats within a subprogram

Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

- Maximum nesting depth for subprograms: 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 L1 *	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

1 Main program UPGMS is executed up to block N170.
2 Subprogram 1 is called, and executed up to block N390.
3 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.
4 Subprogram 1 is executed from block N400 up to block N450. End of subprogram 1 and return jump to the main program SUBPGMS.
5 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 *	Program section between this block and G98 L2
...	(block N200) is repeated twice.
N350 L1.1 *	Program section between this block and G98 L1
...	(block N150) is repeated once.
N999999 %REPS G71 *	

Program execution

1 Main program REPS is executed up to block N270.
2 Program section between block N270 and block N200 is repeated twice.
3 Main program REPS is executed from block N280 to block N350.
4 Program section between block N350 and block N150 is repeated once (including the program section repeat between block N200 and block N270).
5 Main program REPS is executed from block N360 to block N999999 (end of program).

Repeating a subprogram

Example NC blocks

%SUBPGREP G71 *	
...	
N100 G98 L1 *	Beginning of program section repeat 1
N110 L2.0 *	Subprogram call
N120 L1.2 *	Program section between this block and G98 L1
...	(block N100) is repeated twice.
N190 G00 G40 Z+100 M2*	Last block of the main program with M2
N200 G98 L2 *	Beginning of subprogram
...	
N280 G98 L0 *	End of subprogram
N999999 %SUBPGREP G71 *	

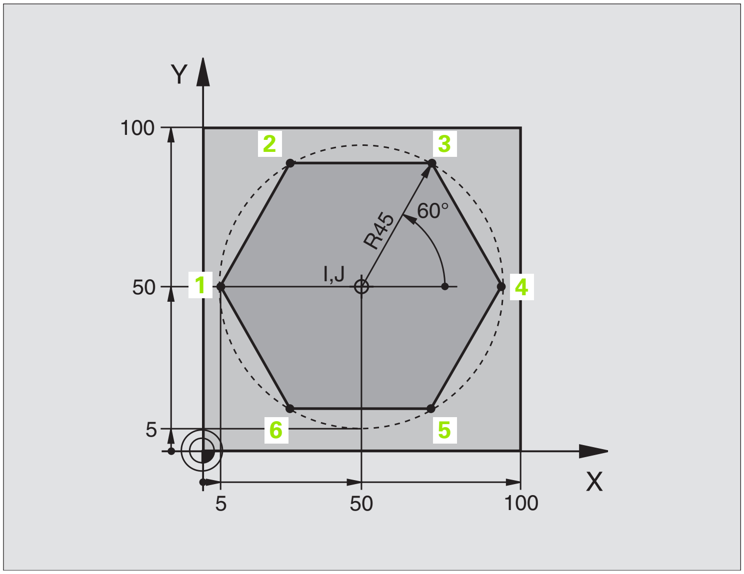
Program execution

1. Main program SUBPGREP is executed up to block N110.
2. Subprogram 2 is called and executed.
3. Program section between block N120 and block N100 is repeated twice. Subprogram 2 is repeated twice.
4. Main program SUBPGREP is executed once 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



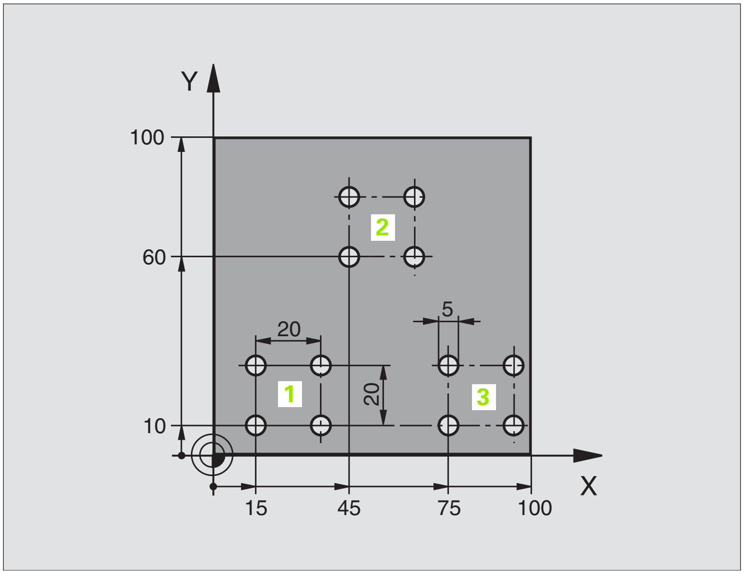
`%PGMWDH G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+7.5 *`	Define the tool
`N40 T1 G17 S4000 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 I+50 J+50 *`	Set pole
`N70 G10 R+60 H+180 *`	Pre-position in the working plane
`N80 G01 Z+0 F1000 M3 *`	Pre-position to the workpiece surface

`N90 G98 L1 *`	Set label for program section repeat
`N100 G91 Z-4 *`	Infeed depth in incremental values (in space)
`N110 G11 G41 G90 R+45 H+180 F250 *`	First contour point
`N120 G26 R5 *`	Approach contour
`N130 H+120 *`	
`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 L1.4 *`	Return jump to label 1; section is repeated a total of 4 times
`N220 G00 Z+250 M2 *`	Retract in the tool axis, end program
`N9999999 %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



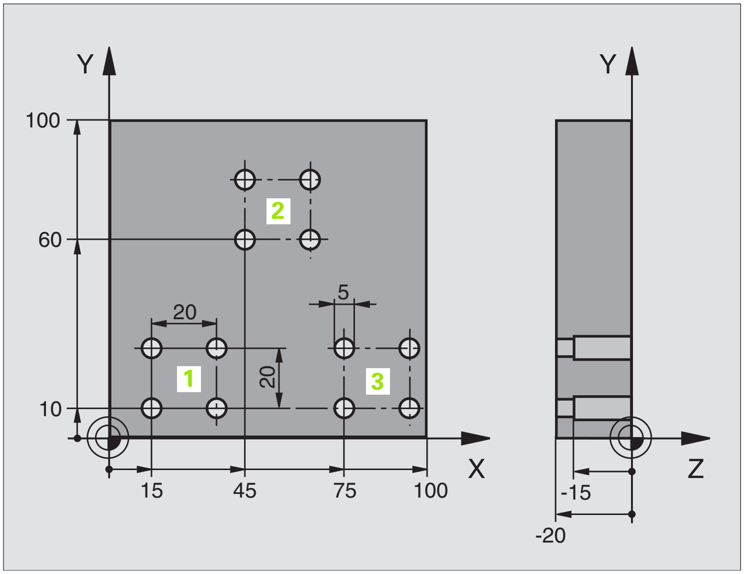
%UP1 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+2.5 *	Define the tool
N40 T1 G17 S5000 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=300 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=2 ;2ND SET-UP CLEARANCE	
Q211=0 ;DWELL TIME AT DEPTH	

N70 X+15 Y+10 M3 *	Move to starting point for group 1
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 G79 *	Call cycle for 1st hole
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
N9999999 %UP1 G71 *	

Example: Group of holes with several tools

Program sequence

- Program the fixed cycles in the main program
- Call the entire hole pattern (subprogram 1)
- Approach the groups of holes in subprogram 1, call group of holes (subprogram 2)
- Program the group of holes only once in subprogram 2



`%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 *`	Define tool: drill
`N50 G99 T3 L+0 R+3.5 *`	Define tool: reamer
`N60 T1 G17 S5000 *`	Call tool: center drill
`N70 G00 G40 G90 Z+250 *`	Retract the tool
`N80 G200 DRILLING`	Cycle definition: Centering
`Q200=2 ;SET-UP CLEARANCE`	
`Q201=-3 ;DEPTH`	
`Q206=250 ;FEED RATE FOR PLNGNG`	
`Q202=3 ;PLUNGING DEPTH`	
`Q210=0 ;DWELL TIME AT TOP`	
`Q203=+0 ;SURFACE COORDINATE`	
`Q204=10 ;2ND SET-UP CLEARANCE`	
`Q211=0.2 ;DWELL TIME AT DEPTH`	
`N90 L1.0 *`	Call subprogram 1 for the entire hole pattern

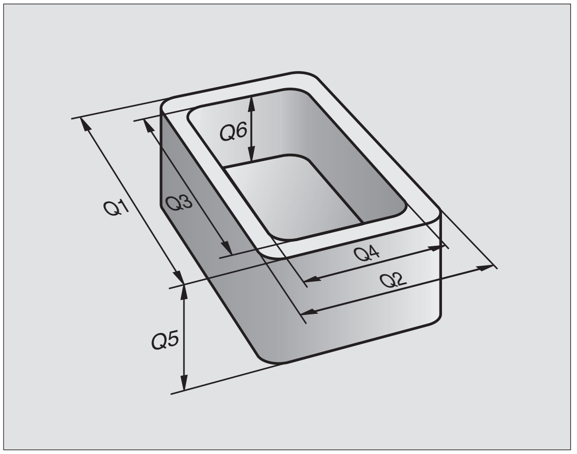
N100 G00 Z+250 M6 *	Tool change
N110 T2 G17 S4000 *	Call toll: drill
N120 D0 Q201 P01 -25 *	New depth for drilling
N130 D0 Q202 P01 +5 *	New plunging depth for drilling
N140 L1.0 *	Call subprogram 1 for the entire hole pattern
N150 G00 Z+250 M6 *	Tool change
N160 T3 G17 S500 *	Call tool: reamer
N80 G201 REAMING	Cycle definition: REAMING
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLUNGING	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
N180 L1.0 *	Call subprogram 1 for the entire hole pattern
N190 G00 Z+250 M2 *	End of main program
N200 G98 L1 *	Beginning of subprogram 1: Entire hole pattern
N210 G00 G40 G90 X+15 Y+10 M3 *	Move to starting point for group 1
N220 L2.0 *	Call subprogram 2 for the group
N230 X+45 Y+60 *	Move to starting point for group 2
N240 L2.0 *	Call subprogram 2 for the group
N250 X+75 Y+10 *	Move to starting point for group 3
N260 L2.0 *	Call subprogram 2 for the group
N270 G98 L0 *	End of subprogram 1
N280 G98 L2 *	Beginning of subprogram 2: Group of holes
N290 G79 *	Call cycle for 1st hole
N300 G91 X+20 M99 *	Move to 2nd hole, call cycle
N310 Y+20 M99 *	Move to 3rd hole, call cycle
N320 X-20 G90 M99 *	Move to 4th hole, call cycle
N330 G98 L0 *	End of subprogram 2
N340 END PGM UP2 MM	

10

Programming: Q Parameters

10.1 Principle and Overview

You can program an entire family of parts in a single part program. You do this by entering variables called Q parameters instead of fixed numerical values.



Q parameters can represent information such as:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

Q parameters also enable you to program contours that are defined with mathematical functions. You can also use Q parameters to make the execution of machining steps depend on logical conditions.

Q parameters are designated by the letter Q and a number between 0 and 299. They are grouped according to three ranges:

Meaning	Range
Freely applicable parameters, globally effective for all programs stored in the TNC memory	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, globally effective for all programs that are stored in the TNC memory	Q200 to Q399

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. Internally, the TNC can calculate up to a width of 57 bits before and 7 bits after the decimal point (32-bit data width corresponds to a decimal value of 4 294 967 296).

Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius (see "Preassigned Q Parameters," page 404).

If you are using the parameters Q60 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

When you are writing a part program, press the “Q” key (in the numeric keypad for numerical input and axis selection, below the +/– key). The TNC then displays the following soft keys:

Function group	Soft key
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
Function for machining complex contours (see “Entering a contour formula,” page 333)	CONTOUR 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

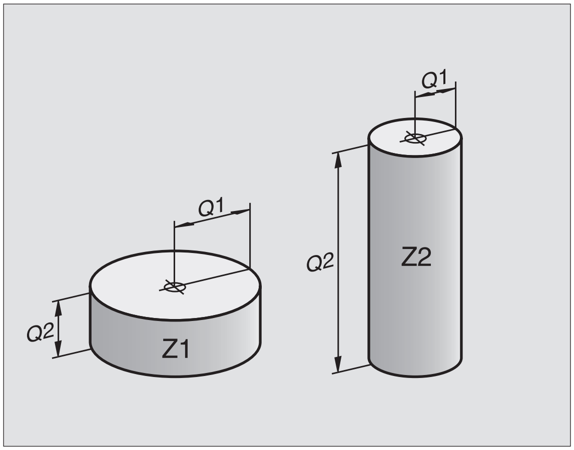
You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

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 Operations

Function

The Q parameters listed below enable you to program basic mathematical functions in a part program:

- Select a Q parameter function: Press the Q key (in the numerical keypad at right). The Q parameter functions are displayed in a soft-key row.
- To select the mathematical functions: Press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

Overview

Function	Soft key
D00: ASSIGN Example: **D00 Q5 P01 +60 *** Assigns a numerical value.	D0 X = Y
D01: ADDITION Example: **D01 Q1 P01 -Q2 P02 -5 *** Calculates and assigns the sum of two values.	D1 X + Y
D02: SUBTRACTION Example: **D02 Q1 P01 +10 P02 +5 *** Calculate and assign the difference of two values	D2 X - Y
D03: MULTIPLICATION Example: **D03 Q2 P01 +3 P02 +3 *** Calculate and assign the product of two values	D3 X * Y
D04: DIVISION Example: **D04 Q4 P01 +8 P02 +Q2 *** Calculates and assigns the quotient of two values. Not permitted: division by 0	D4 X / Y
D05: SQUARE ROOT Example: **D05 Q50 P01 4 *** Calculates and assigns the square root of a number. **Not permitted:** Square root of a negative number	D5 SQRT

To the right of the "=" character you can enter the following:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming fundamental operations

Programming example 1:

Q	Call the Q parameter functions by pressing the Q 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 NO. FOR RESULT?

5 ENT	Enter the number of the Q parameter, e.g. 5.

1. VALUE OR PARAMETER?

10 ENT	Assign the value 10 to Q5.

Example: NC block

```
N16 D00 P01 +10 *
```

Programming example 2:

Q	Call the Q parameter functions by pressing the Q 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 NO. FOR RESULT?

12 ENT	Enter the number of the Q parameter, e.g. 12.

1. VALUE OR PARAMETER?

Q5 ENT	Enter Q5 for the first value.

2. VALUE OR PARAMETER?

7 ENT	Enter 7 for the second value.

Example: NC block

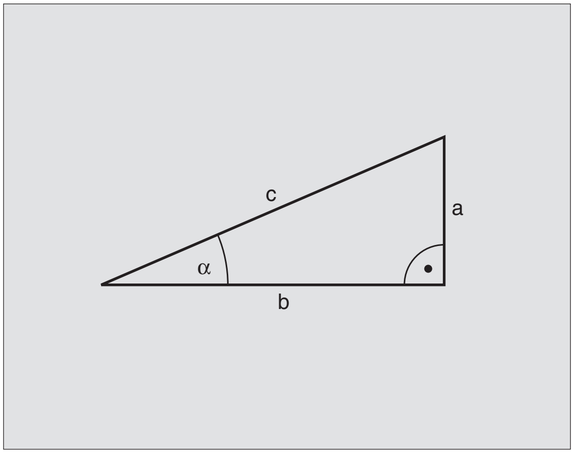
```
N17 D03 Q12 P01 +Q5 P02 +7 *
```

10.4 Trigonometric Functions

Definitions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. In this case:

Sine: $\sin \alpha = a / c$
Cosine: $\cos \alpha = b / c$
Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$



where

- c is the side opposite the right angle
- a is the side opposite the angle *a*
- b is the third side.

The TNC can find the angle from the tangent

$\alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

Example:

a = 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 below.

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

Function

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels, see "Labeling Subprograms and Program Section Repeats," page 372). 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:

D09 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 Y 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 Y GOTO
D11: IF GREATER THAN, 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 Y GOTO
D12: IF LESS THAN, 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 Y 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

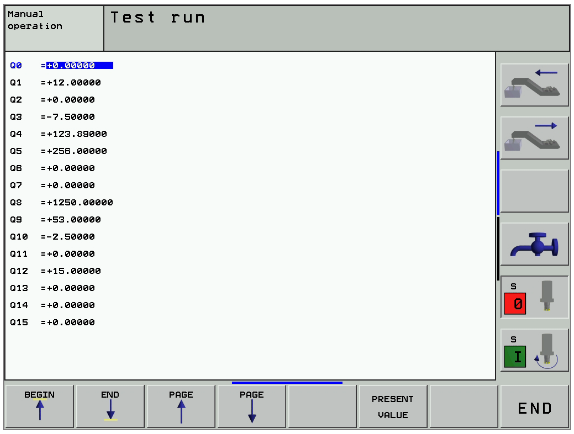
Procedure

You can check and edit Q parameters when writing, testing and running programs in the Programming and Editing, Test Run, Program Run Full Sequence, and Program Run Single Block modes.

- If you are in a program run, interrupt it if required (for example by pressing the machine STOP button and the INTERNAL STOP soft key). If you are in a test run, interrupt it.

Q

- To call Q parameter functions: Press the Q key or the Q INFO soft key in the Programming and Editing mode of operation.
- The TNC lists all parameters and their current values. With the arrow keys or the soft keys, go pagewise to the desired parameters.
- If you would like to change the value, enter a new value and confirm with the ENT key.
- To leave the value unchanged, press the PRESENT VALUE soft key or end the dialog with the END key.



The parameters (parameter numbers > 100) used by the TNC are provided with comments.

10.7 Additional Functions

Overview

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key
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

Example NC block

The TNC is to display the text stored under error number 254.

```
N180 D14 P01 254 *
```

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 D 14 during program run, it will interrupt the run and display an error message. The program must then be restarted. The error numbers are listed in the table below.

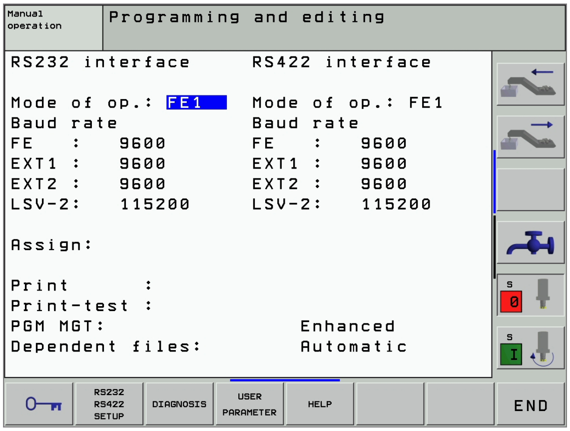
Range of error numbers	Standard dialog text
0 ... 299	D 14: Error number 0 299
300 ... 999	Machine-dependent dialog
1000 ... 1099	Internal error messages (see table at right)

Error number	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 small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Enter Q218 greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Enter Q222 greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be < 360°
1040	Enter Q223 greater than Q222
1041	Q214: 0 not permitted

Error number	Text
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter direction Q351 unequal 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as a negative value
1078	Q303 not defined in measuring cycle
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory measuring points

D15: PRINT: Output of texts or Q parameter values

> Setting the data interface: In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters (see “Assign,” page 443).



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 %FN 15RUN.A (output in program run mode) or in the file %FN15SIM.A (output in test run mode). The data are transmitted from a buffer. Data output begins at the latest by program end or when you stop the program. In the Single Block mode of operation, data transfer begins at block end.

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

```
N67 D15 P01 20 *
```

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

```
N56 D19 P01 +10 P02 +Q3 *
```

10.8 Entering Formulas Directly

Entering formulas

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Press the FORMULA soft key to call the formula functions. The TNC displays the following soft keys in several soft-key rows:

Logic command	Soft key
Addition Example: **Q10 = Q1 + Q5**	+
Subtraction Example: **Q25 = Q7 – Q108**	-
Multiplication Example: **Q12 = 5 * Q5**	*
Division Example: **Q25 = Q1 / Q2**	/
Opening parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	(
Closing parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	)
Square of a value Example: **Q15 = SQ 5**	SQ
Square root Example: **Q22 = SQRT 25**	SQRT
Sine of an angle Example: **Q44 = SIN 45**	SIN
Cosine of an angle Example: **Q45 = COS 45**	COS
Tangent of an angle Example: **Q46 = TAN 45**	TAN
Arc sine Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: **Q10 = ASIN 0.75**	ASIN
Arc cosine Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: **Q11 = ACOS Q40**	ACOS

Logic command	Soft key
Arc tangent Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: **`Q12 = ATAN Q50`**	ATAN
Powers of values Example: **`Q15 = 3^3`**	^
Constant "pi" (3.14159) Example: **`Q15 = PI`**	PI
Natural logarithm (LN) of a number Base 2.7183 Example: **`Q15 = LN Q11`**	LN
Logarithm of a number, base 10 Example: **`Q33 = LOG Q22`**	LOG
Exponential function, 2.7183 to the power of n Example: **`Q1 = EXP Q12`**	EXP
Negate (multiplication by -1) Example: **`Q2 = NEG Q1`**	NEG
Truncate decimal places Form an integer Example: **`Q3 = INT Q42`**	INT
Absolute value of a number Example: **`Q4 = ABS Q22`**	ABS
Truncate places before the decimal point Form a fraction Example: **`Q5 = FRAC Q23`**	FRAC
Check algebraic sign of a number Example: **`Q12 = SGN Q50`** If result for Q12 = 1: Q50 >= 0 If result for Q12 = 0: Q50 < 0	SGN
Calculate modulo value Example: **`Q12 = 400 % 360`** Result: Q12 = 40	%

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first (multiplication and division before addition and subtraction)

```
N112  Q1 = 5 * 3 + 2 * 10 = 35
```

1st calculation: 5 * 3 = 15
2nd calculation: 2 * 10 = 20
3rd calculation: 15 +20 = 35

or

```
N113  Q2 = SQ 10 - 3^3 = 73
```

1st calculation: 10 squared = 100
2nd calculation: 3 to the power of 3 = 27
3rd calculation: 100 – 27 = 73

Distributive law
for calculating with parentheses

a * (b + c) = a * b + a * c

Programming example

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.

Q	Call the Q parameter functions by pressing the Q key.
FORMULA	For formula input, press the FORMULA soft key.

PARAMETER NO. FOR RESULT?

ENT 25	Enter the parameter number.
▷ ATAN	Shift the soft-key row and select the arc tangent function.
◁ (	Shift the soft-key row and open the parentheses.
Q 12	Enter Q parameter number 12.
/	Select division.
Q 13	Enter Q parameter number 13.
) END	Close parentheses and conclude formula entry.

Example NC block

```
N37  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
- Delta value DR from the TOOL CALL block

Tool axis: Q109

The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = -1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

Spindle status: Q110

The value of Q110 depends on which M function was last programmed for the spindle:

M Function	Parameter value
No spindle status defined	Q110 = -1
M03: Spindle ON, clockwise	Q110 = 0
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M Function	Parameter value
M08: Coolant ON	Q111 = 1
M09: Coolant OFF	Q111 = 0

Overlap factor: Q112

The overlap factor for pocket milling (MP7430) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

The value of parameter Q113 specifies whether the highest-level NC program (for nesting with %...) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe. The coordinates are referenced to the datum that is currently active in the Manual operating mode.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis dependent on MP100	Q118
Vth axis dependent on MP100	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 130

Actual-nominal deviation	Parameter value
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles: Rotary axis coordinates calculated by the TNC

coordinates	Parameter value
A axis	Q120
B axis	Q121
C axis	Q122

Results of measurements with touch probe cycles

(also see the Touch Probe Cycles User’s Manual)

Uncorrected coordinates of the last touch point	Parameter value
Reference axis	Q141
Minor axis	Q142
Touch probe axis	Q143

Measured actual values	Parameter value
Angle of a straight line	Q150
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

Determined deviation	Parameter value
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

Determined solid angles	Parameter value
Rotation about the A axis	Q170
Rotation about the B axis	Q171
Rotation about the C axis	Q172

Workpiece status	Parameter value
Good	Q180
Re-work	Q181
Scrap	Q182

Measured deviation with cycle 440	Parameter value
X axis	Q185
Y axis	Q186
Z axis	Q187

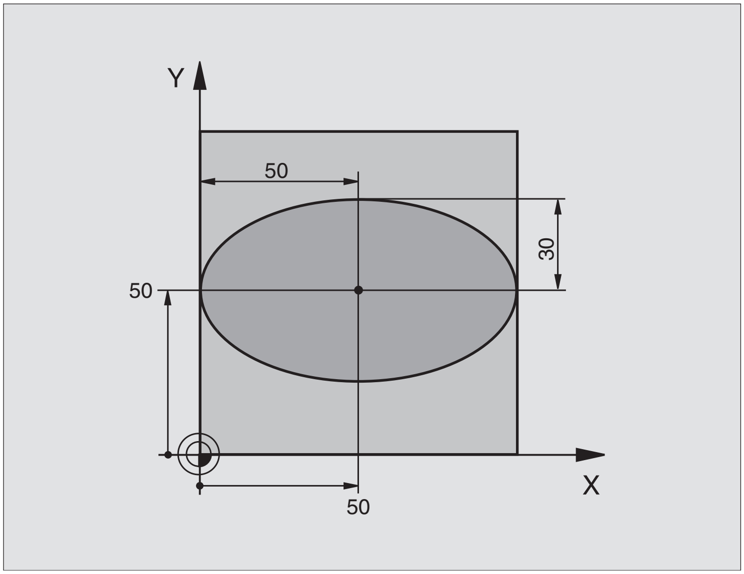
Reserved for internal use	Parameter value
Markers for cycles (point patterns)	Q197
Number of the active touch probe cycle	Q198

Status during tool measurement with TT	Parameter value
Tool within tolerance	Q199 = 0.0
Tool is worn (LTOL/RTOL exceeded)	Q199 = 1.0
Tool is broken (LBREAK/RBREAK exceeded)	Q199 = 2.0

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculation steps you define for the lines, the smoother the curve becomes.
- The machining direction can be altered by changing the entries for the starting and end angles in the plane:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction: starting angle < end angle
- The tool radius is not taken into account.



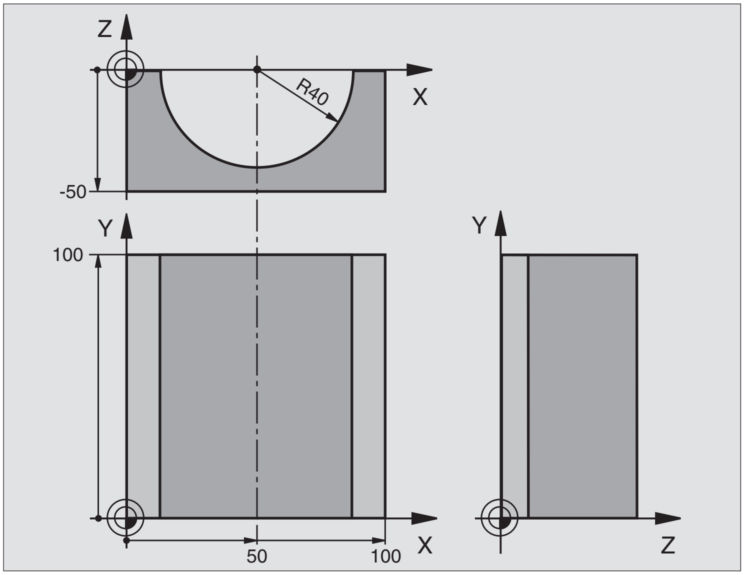
%ELLIPSE G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q3 P01 +50 *	Semiaxis in X
N40 D00 Q4 P01 +30 *	Semiaxis in Y
N50 D00 Q5 P01 +0 *	Starting angle in the plane
N60 D00 Q6 P01 +360 *	End angle in the plane
N70 D00 Q7 P01 +40 *	Number of calculation steps
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 *	Set-up 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 *	Tool call
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 set-up 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 set-up clearance
`N410 G98 L0 *`	End of subprogram
`N999999 %ELLIPSE 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.



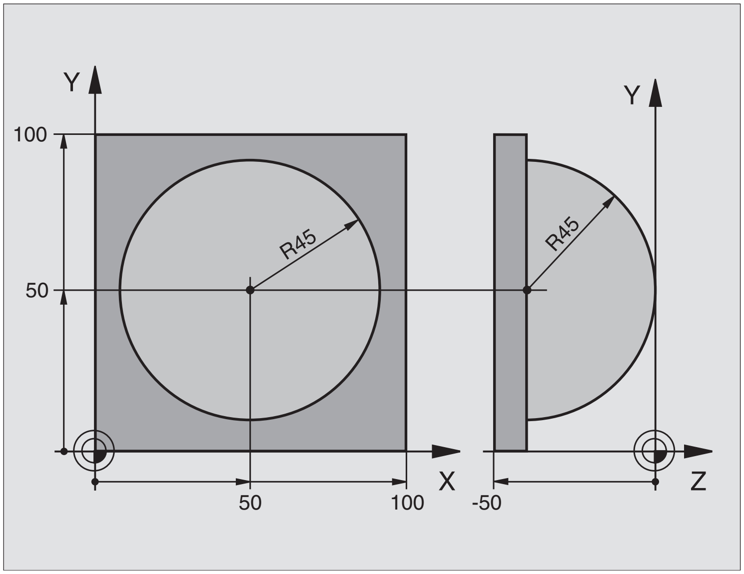
%CYLIN G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +0 *	Center in Y axis
N30 D00 Q3 P01 +0 *	Center in Z axis
N40 D00 Q4 P01 +90 *	Starting angle in space (Z/X plane)
N50 D00 Q5 P01 +270 *	End angle in space (Z/X plane)
N60 D00 Q6 P01 +40 *	Radius of the cylinder
N70 D00 Q7 P01 +100 *	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 *	Tool call
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 G98 L1 *`	
`N320 I+0 K+0 *`	Set pole in the Z/X plane
`N330 G11 R+Q16 H+Q24 FQ11 *`	Move to starting position on cylinder, plunge-cutting obliquely into the material
`N340 G01 G40 Y+Q7 FQ12 *`	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 FQ11 *`	Move in an approximated “arc” for the next longitudinal cut
`N390 G01 G40 Y+0 FQ12 *`	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 LBL 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 in Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically.



%SPHERE 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 *	Set-up 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 *	Tool call
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 G98 L1 *	Pre-position in the tool axis
N320 I+0 J+0 *	Set pole in the X/Y plane for pre-positioning
N330 G11 G40 R+Q26 H+Q8 FQ12 *	Pre-position in the plane
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 LBL 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 %SPHERE G71 *	

11

Test Run and Program Run

11.1 Graphics

Function

In the Program Run modes of operation as well as in the Test Run mode, the TNC provides the following display modes. Using soft keys, select whether you desire:

- Plan view
- Projection in 3 planes
- 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter. For this purpose, enter R2 = R in the tool table.

The TNC will not show a graphic if

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

With Machine Parameters 7315 to 7317 you can 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 rotary axis movements or a tilted working plane are defined. In this case, the TNC will display an error message.
>
> The TNC graphic does not show a radius oversize **DR** that has been programmed in the **T** block.

Overview of display modes

The TNC displays the following soft keys in the Program Run and Test Run modes of operation:

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

Limitations during program run

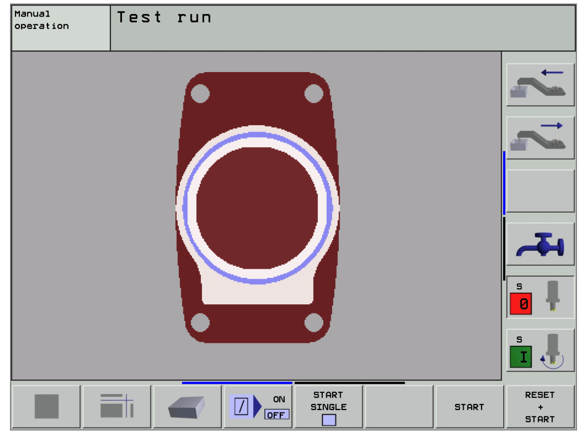
A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text **ERROR** in the graphics window. The machining process is continued, however.

Plan view

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

▶ Press the soft key for plan view.

▶ Regarding depth display, remember: The deeper the surface, the darker the shade.

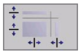


Projection in 3 planes

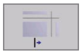
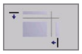
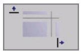
Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

Details can be isolated in this display mode for magnification (see “Magnifying details,” page 420).

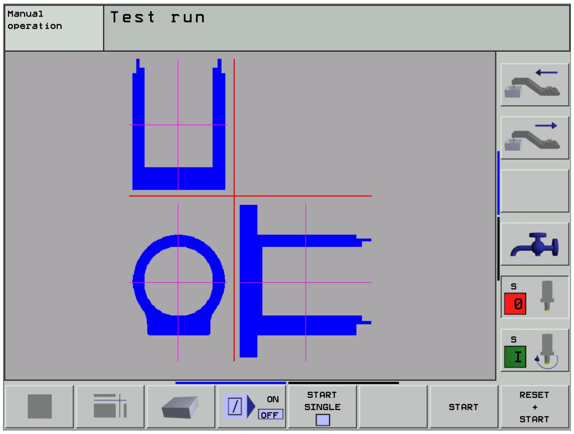
In addition, you can shift the sectional planes with the corresponding soft keys:



- Select the soft key for projection in three planes.
- Shift the soft-key row and select the soft key for sectional planes.
- The TNC then displays the following soft keys:

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

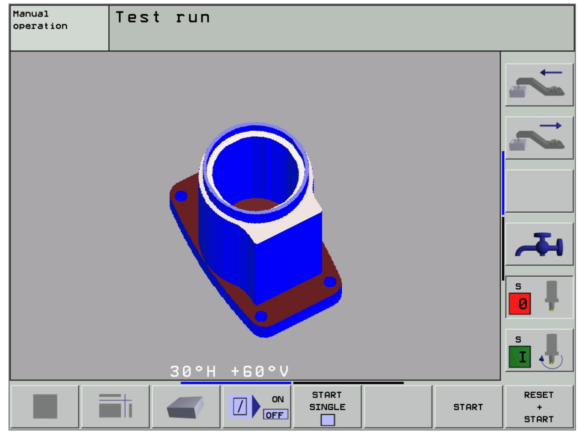
The positions of the sectional planes are visible during shifting.



Coordinates of the line of intersection

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 MP7310.

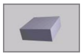
3-D view



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

You can rotate the 3-D display about the vertical and horizontal axes. The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation.

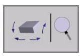
In the Test Run mode of operation you can isolate details for magnification, see "Magnifying details," page 420.



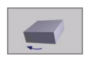
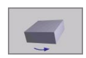
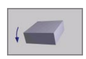
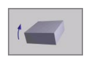
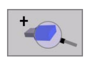
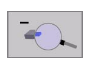
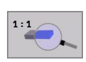
- Press the soft key for 3-D view.

Rotating and magnifying/reducing the 3-D view

- Shift the soft-key row until the soft key for the rotating and magnification/reduction appears.

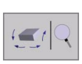


- Select functions for rotating and magnifying/reducing:

Function	Soft keys	
Rotate in 5° steps about the vertical axis		
Rotate in 5° steps about the horizontal axis		
Magnify the graphic stepwise. If the view is magnified, the TNC shows the letter **Z** in the footer of the graphic window.		
Reduce the graphic stepwise. . If the view is magnified, the TNC shows the letter **Z** in the footer of the graphic window.		
Reset image to programmed size		

Switch the frame overlay display for the workpiece blank on/off:

- Shift the soft-key row until the soft key for the rotating and magnification/reduction appears.

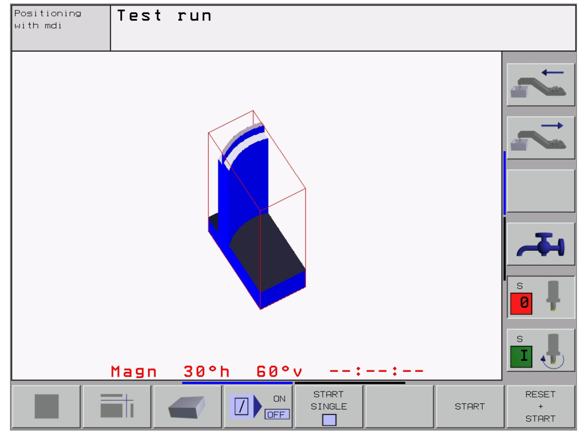


- Select functions for rotating and magnifying/reducing:

- Show the frame for the BLK FORM: Set the highlight in the soft key to SHOW

- Hide the frame for the BLK FORM: Set the highlight in the soft key to OMIT

Magnifying details



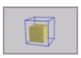
You can magnify details in all display modes in the Test Run mode and a program run mode.

The graphic simulation or the program run, respectively, must first have been stopped. A detail magnification is always effective in all display modes.

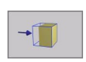
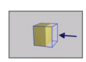
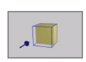
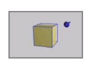
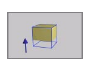
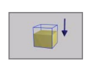
Changing the detail magnification

The soft keys are listed in the table.

- Interrupt the graphic simulation, if necessary.
- Shift the soft-key row in the Test Run mode, or in a program run mode, respectively, until the soft key for detail enlargement appears.



- Select the functions for section magnification.
- Press the corresponding soft key to select the workpiece surface (see table below).
- To reduce or magnify the blank form, press and hold the MINUS or PLUS soft key, respectively.
- Restart the test run or program run by pressing the START soft key (RESET + START returns the workpiece blank to its original state).

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	

Cursor position during detail magnification

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 indicated 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, reduce or enlarge 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 the blank form was programmed.	WINDOW BLK FORM

With the WINDOW BLK FORM soft key, you return the displayed workpiece blank to its originally programmed dimensions, even after isolating a detail—without TRANSFER DETAIL.

Measuring the machining time

Program Run modes of operation

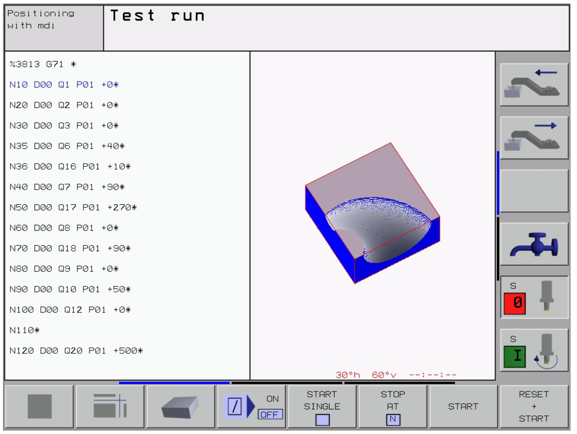
The timer counts and displays the time from program start to program end. The timer stops whenever machining is interrupted.

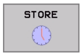
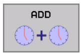
Test Run

The timer displays the approximate time that 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.

Activating the stopwatch function

Shift the soft-key rows until the TNC displays the following soft keys with the stopwatch functions:



Stopwatch functions	Soft key
Store displayed time	
Display the sum of stored time and displayed time	
Clear displayed time	

The soft keys available to the left of the stopwatch functions depend on the selected screen layout.

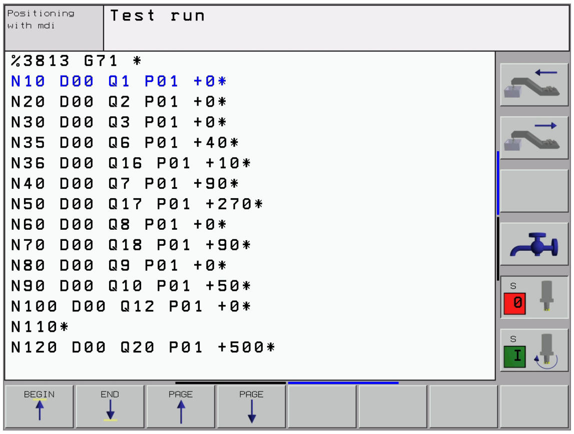
The time is reset when a new BLK form is entered.

11.2 Functions for Program Display

Overview

In the program run modes of operation as well as in the Test Run mode, the TNC provides the following soft keys for displaying a part program in pages:

Function	Soft key
Go back in the program by one screen	PAGE ↑
Go forward in the program by one screen	PAGE ↓
Go to beginning of program	BEGIN ↑
Go to end of program	END ↓



11.3 Test Run

Function

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

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

The following functions are also available:

- Blockwise test run
- 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 work space monitoring for the test run (see "Showing the Workpiece in the Working Space," page 453).

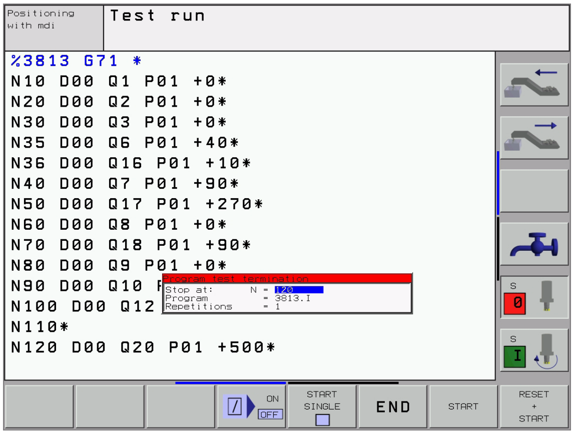
- Select the Test Run operating mode
- Call the file manager with the PGM MGT key and select the file you wish to test, or
- Go to the program beginning: Select line "0" with the GOTO key and confirm your entry with the ENT key.

The TNC then displays the following soft keys:

Function	Soft key
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

Run 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 run a program test up to a specific block, press the STOP AT N soft key.

- **Stop at N:** Enter the block number at which you wish the test to stop.
- **Program:** Enter the name of the program that contains the block with the selected block number. The TNC displays the name of the selected program. If the test run is to be interrupted in a program that was called with %, you must enter this name.
- **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

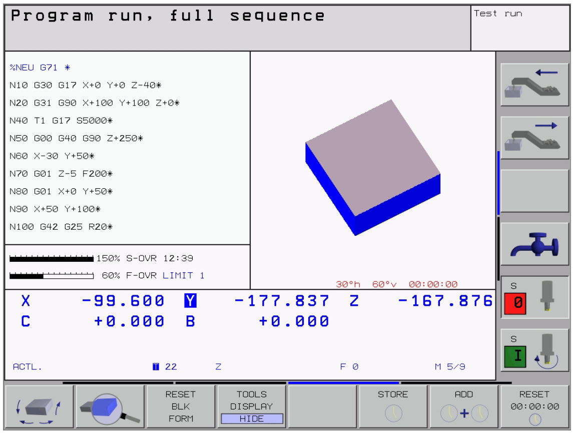
Function

In the Program Run, Full Sequence mode 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
- Functions for graphic simulation
- Additional status display



Running a part program

Preparation

1 Clamp the workpiece to the machine table.

2 Set the datum.

3 Select the necessary tables and pallet files (status M).

4 Select the part program (status M).

> You can adjust the feed rate and spindle speed with the override knobs.
>
> It is possible to reduce the rapid traverse speed when starting the NC program using the FMAX soft key. The entered value remains in effect even after the machine has been turned off and on again. In order to re-establish the original rapid traverse speed, you need to re-enter the corresponding value.

Program Run, Full Sequence

- Start the part program with the machine START button.

Program Run, Single Block

- Start each block of the part program individually with the machine START button.

Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- 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)

Interrupting the machining process 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.

Interrupting the machining process 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 the Program Run, Single Block mode. 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.

Danger 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 returning to the 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.

Application example:
Retracting the spindle after tool breakage

- Interrupt machining.
- Enable the external direction keys: Press the MANUAL OPERATION soft key.
- Move the axes with the machine axis direction buttons.

On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Refer to your machine manual.

Resuming program run after an interruption

> If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.
>
> If you interrupt a program run during execution of a subprogram or program section repeat, use the RESTORE POS AT 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 (e.g. datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

> Note that the stored data remain active until they are reset (e.g. if you select a new program).

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION soft key).

Resuming program run with the START button

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

- The machine STOP button was pressed.
- An interruption was programmed.

Resuming program run after an error

If the error message is not blinking:

- ▶ Remove the cause of the error.
- ▶ To clear the error message from the screen, press the CE key.
- ▶ Restart the program, or resume program run where it was interrupted.

If 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. Refer to your machine manual.

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.

All necessary programs, tables and pallet files must be selected in a Program Run mode of operation (status M).

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan.

After a block scan, return the tool to the calculated position with RESTORE POSITION.

Tool length compensation does not take effect until after the tool call and a following positioning block; this also applies to an altered tool length.

If you are working with nested programs, you can use MP7680 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.

The function M128 is not permitted during a mid-program startup.

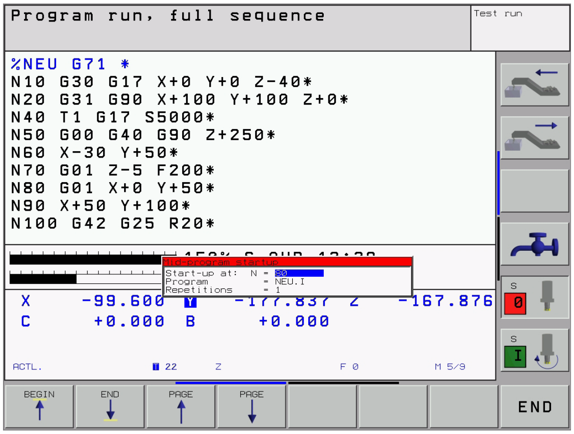
If the working plane is tilted, you can use the 3-D ON/OFF soft key to define whether the TNC is to return to the contour in a tilted or in a non-tilted coordinate system.

If you want to use the block scan feature in a pallet table, select the program in which a mid-program startup is to be performed from the pallet table by using the arrow keys. Then press the RESTORE POS AT N soft key.

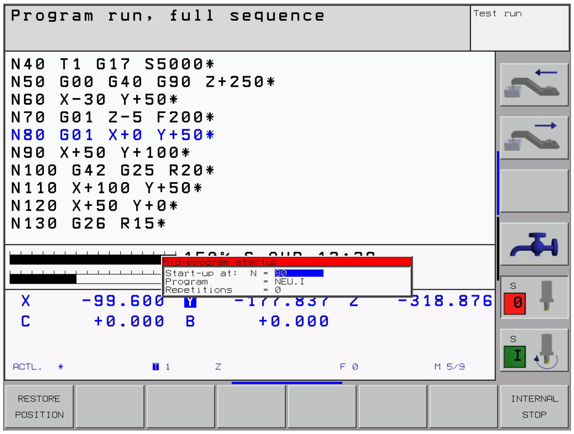
All touch probe cycles and Cycle 247 are skipped in a mid-program startup. Result parameters that are written to from these cycles might therefore remain empty.

- To go to the first block of the current program to start a block scan, enter GOTO "0".
- To select mid-program startup, press the RESTORE POS AT N soft key.

- **`Start-up at N:`** Enter the block number N at which the block scan should end.
- **`Program:`** Enter the name of the program containing block N.
- **`Repetitions:`** If block N is located in a program section repeat, enter the number of repetitions to be calculated in the block scan.
- PLC ON/OFF: 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. The tool in the spindle must equal the tool called by the program.
- To start the block scan, press the machine START button.
- To return to the contour, see “Returning to the contour,” page 433



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.
- 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.

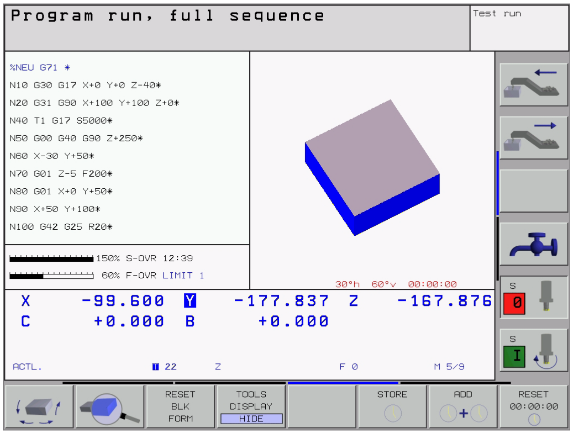
▶ To resume machining, press the machine START key.

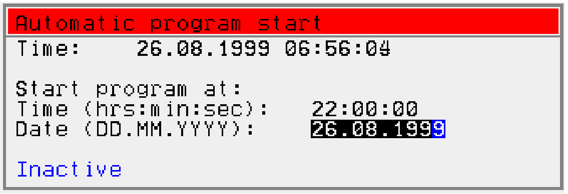
11.5 Automatic Program Start

Function

The TNC must be specially prepared by the machine tool builder for use of the automatic program start function. Refer to your machine manual.

In a Program Run operating mode, you can use the soft key AUTOSTART (see figure at upper right) to define a specific time at which the program that is currently active in this operating mode is to be started:





- Show the window for entering the starting time (see figure at center right).
- **Time (h:min:sec):** Time of day at which the program is to be started.
- **Date (DD.MM.YYYY):** Date at which the program is to be started.
- To activate the start, set the AUTOSTART soft key to ON.

11.6 Optional block skip

Function

In a test run or program run, the TNC can skip over blocks that begin with a slash “/”:

- To run or test the program without the blocks preceded by a slash, set the soft key to ON.

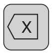
- To run or test the program with the blocks preceded by a slash, set the soft key to OFF.

This function does not work for G99 blocks.

After a power interruption the control returns to the most recently selected setting.

Erasing the “/” character

- In the **Programming and Editing** mode you select the block in which the character is to be erased.



- Erase the “/” character.

11.7 Optional Program Run Interruption

Function

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.

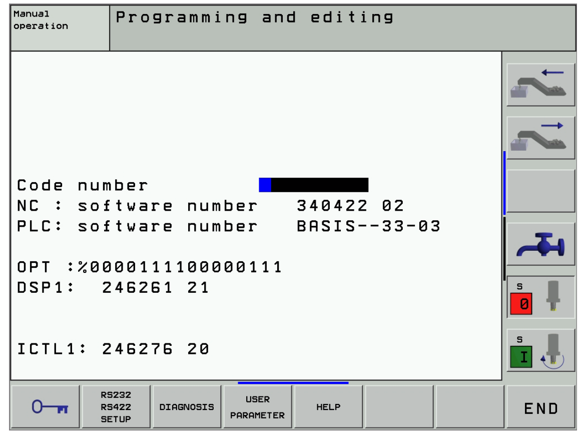
- Interrupt Program Run or Test Run at blocks containing M01: Set soft key to ON.

12

MOD Functions

12.1 MOD functions

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

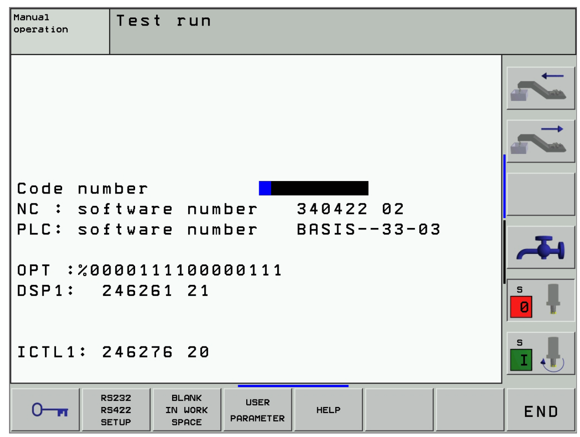


Selecting the MOD functions

Call the operating mode in which you wish to change the MOD functions.

MOD

▶ Press the MOD key. Select the MOD functions for programming/editing and test run. Figures at right, figure on next page: MOD function in a machine operating mode.



Changing the settings

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

There are three possibilities for changing a setting, depending on the function selected:

- Enter 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. If more than one possibility is available for a particular setting, you can superimpose a window listing all of the given possibilities by pressing the GOTO key. Select the desired setting directly by pressing the corresponding numerical key (to the left of the colon), or by using the arrow keys and then confirming with ENT. If you don't want to change the setting, close the window again with END.

Exiting the MOD functions

▶ Close the MOD functions with the END key or soft key.

Overview of MOD functions

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

Programming and Editing:

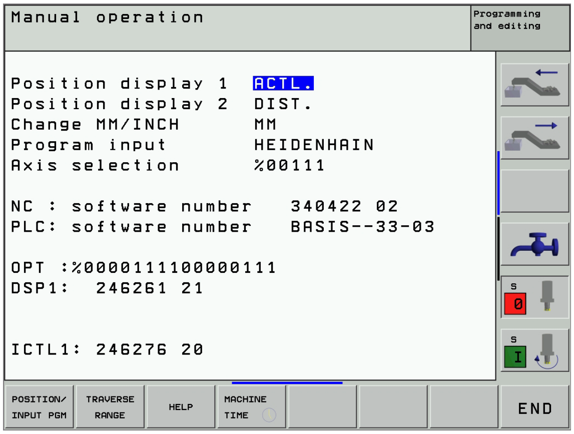
- Display software numbers
- Enter code number
- Set data interface
- Machine-specific user parameters (if provided)
- HELP files (if provided)

Test Run:

- Display software numbers
- Enter code number
- Setting the data interface
- Showing the Workpiece in the Working Space
- Machine-specific user parameters (if provided)
- Displaying HELP files (if provided)

In all other modes:

- Display software numbers
- 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
- Set the axis traverse limits
- Display the datums
- Display operating times
- HELP files (if provided)



12.2 Software Numbers and Option Numbers

Function

The following software numbers are displayed on the TNC screen after the MOD functions have been selected:

- **NC:** Number of the NC software (managed by HEIDENHAIN)
- **PLC:** Number and name of the PLC software (managed by your machine tool builder)
- **SETUP:** Number of the cycle software and the used soft keys (managed by HEIDENHAIN)
- **DSP1:** Number of the speed controller software (managed by HEIDENHAIN)
- **ICTL1:** Number of the current controller software (managed by HEIDENHAIN)

In addition, coded numbers for the options available on your control are displayed after the abbreviation **OPT:**

No options active	%0000000000000000
Bit 0 to bit 7: Additional control loops	%00000000**00000011**
Bit 8 to bit 15: Software options	%**00000011**00000011

12.3 Code Numbers

Function

Code numbers allow you to access various functions that are not always required for normal operation of the TNC.

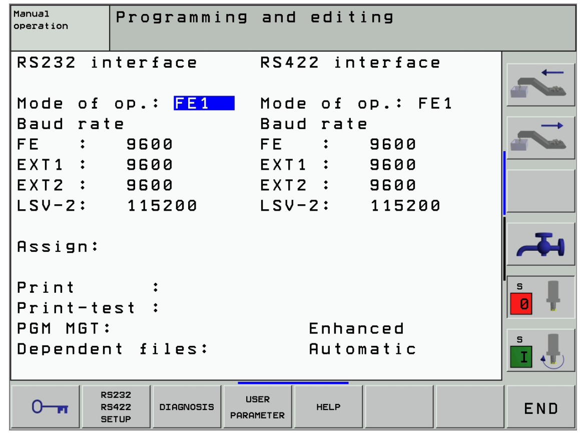
The TNC requires a code number for the following functions:

Function	Code number
Select user parameters	123
Enable special functions for Q-parameter programming	555343
Configuring an Ethernet card	NET123

12.4 Setting the Data Interfaces

Function

To set up the data interfaces, press the RS-232 / RS-422 SETUP soft key 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
PC with HEIDENHAIN software TNCremo for remote operation of the TNC	LSV2	
PC with HEIDENHAIN data transfer software TNCremo	FE1	
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	
Non-HEIDENHAIN devices such as punchers, PC without TNCremo	EXT1, EXT2	

Assign

This function sets the destination for the transferred data.

Applications:

- Transferring values with Q parameter function D15

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 the data in the same directory as the program with D15.	- vacant -

File names

Data	Operating mode	File name
Values with D15	Program Run	%FN15RUN.A
Values with D15	Test Run	%FN15SIM.A

Software for data transfer

For transfer of files to and from the TNC, we recommend using one of the HEIDENHAIN TNCremo data transfer software products for data transfer, such as TNCremo or TNCremoNT. With TNCremo/ TNCremoNT, data transfer is possible with all HEIDENHAIN controls via serial interface.

> Please contact your HEIDENHAIN agent if you would like to receive the TNCremo or TNCremoNT data transfer software.

System requirements for TNCremo:

- AT personal computer or compatible system
- Operating system MS-DOS/PC-DOS 3.00 or later, Windows 3.1, Windows for Workgroups 3.11, Windows NT 3.51, OS/2
- 640 KB working memory
- 1 MB free memory space on your hard disk
- One free serial interface
- A Microsoft-compatible mouse (for ease of operation, not essential)

System requirements for TNCremoNT:

- PC with 486 processor or higher
- Operating system Windows 95, Windows 98, Windows NT 4.0, Windows 2000
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

Installation under Windows

- Start the SETUP.EXE installation program with the file manager (Explorer).
- Follow the setup program instructions.

Starting TNCremo under Windows 3.1, 3.11 and NT 3.51

Windows 3.1, 3.11, NT 3.51:

- Double-click on the icon in the program group HEIDENHAIN Applications.

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.

Starting TNCremoNT under Windows 95, Windows 98 and NT 4.0

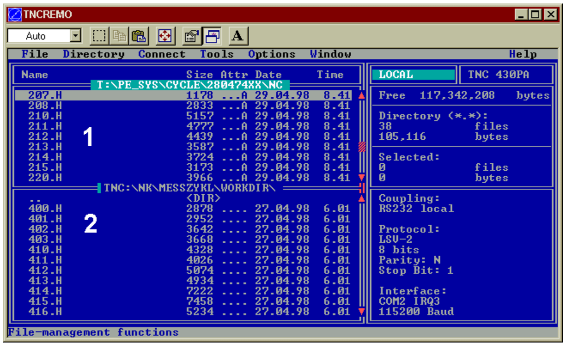
- Click <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremoNT>

When you start TNCremoNT for the first time, TNCremoNT automatically tries to set up a connection with the TNC.

Data transfer between the TNC and TNCremo

Ensure that:

- The TNC is connected to the correct serial port on your PC.
- The operating mode of the interface is set to **LSV2** on the TNC.
- 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 side of the main window **1** . Using the menu items <Directory>, <Change>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- Select <Connect>, <Link>. 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 (highlighted with a mouse click) and activate the functions <File> <Transfer>.

If you want to control data transfer from the TNC, establish the connection with your PC in the following manner:

- Select <Connect>, <File server (LSV2)>. TNCremo is now in server mode. It can receive data from the TNC and send data to the TNC.
- You can now call the file management functions on the TNC by pressing the key PGM MGT (see “Data transfer to or from an external data medium” on page 74) and transfer the desired files.

End TNCremo

Select the menu items <File>, <Exit>, or press the key combination ALT+X.

Refer also to the TNCremoNT help texts where all of the functions are explained in more detail.

Data transfer between the TNC and TNCremoNT

Ensure that:

- The TNC is connected to the correct serial port on your PC or to the network, respectively.
- The operating mode of the interface is set to **LSV2** on the TNC.

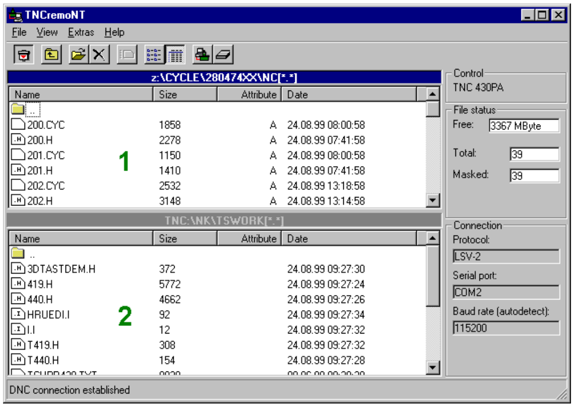
Once you have started TNCremoNT, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using the menu items <File> and <Change directory>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- Select <File>, <Setup connection>. TNCremoNT now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**.
- To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**.
- To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**.

If you want to control data transfer from the TNC, establish the connection with your PC in the following manner:

- Select <Extras>, <TNCserver>. TNCremoNT is now in server mode. It can receive data from the TNC and send data to the TNC.
- You can now call the file management functions on the TNC by pressing the key PGM MGT (see “Data transfer to or from an external data medium” on page 74) and transfer the desired files.



End TNCremoNT

Select the menu items <File>, <Exit>.

> Refer also to the TNCremoNT help texts where all of the functions are explained in more detail.

12.5 Ethernet Interface

Introduction

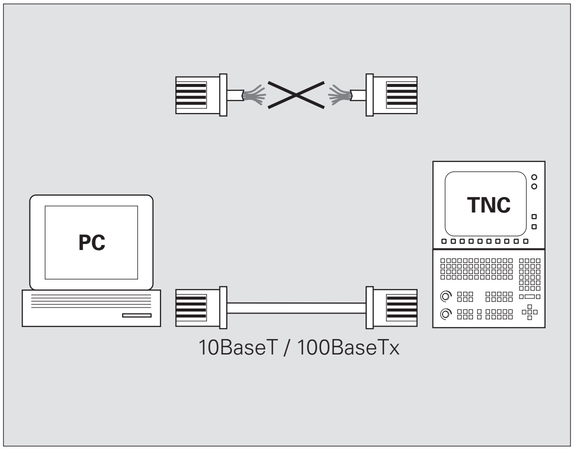
The TNC is shipped with a standard Ethernet card to connect the control as a client in your network. The TNC transmits data through the Ethernet card in accordance with the Transmission Control Protocol/Internet Protocol (TCP/IP) family of protocols and with the aid of the Network File System (NFS).

Connection possibilities

You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 100BaseTX). The connection is metallically isolated from the control electronics.

RJ45 connection X26 (100BaseTX or 10BaseT)

For a 100BaseTX or 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.



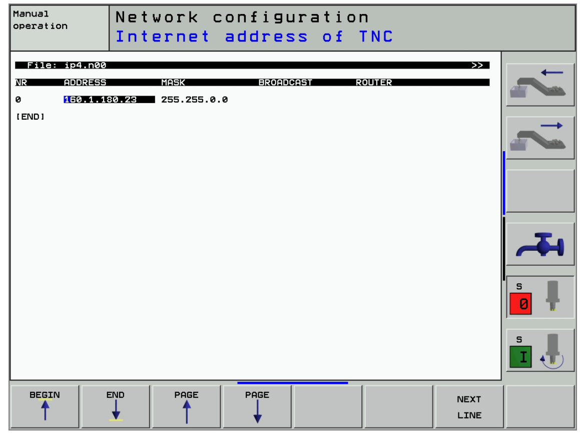
The maximum cable length between TNC and a node depends on the quality grade of the cable, the sheathing and the type of network (100BaseTX or 10BaseT).

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 keyword NET123. The TNC will then display the main screen for network configuration.



General network settings

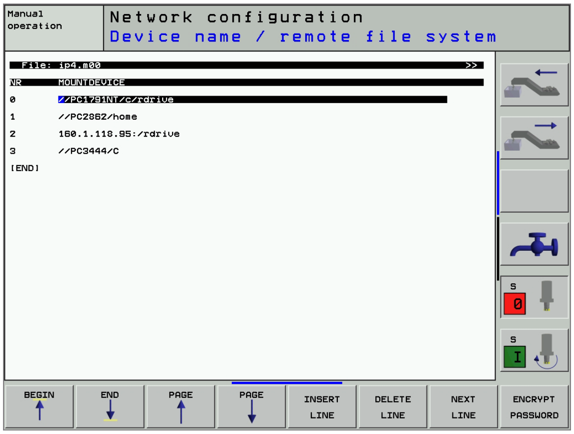
- Press the DEFINE NET soft key to enter the general network settings and enter the following information:

Setting	Meaning
ADDRESS	Address that your network specialist must assign to the TNC. Input: four numerical values separated by points, e.g. 160.1.180.20
MASK	The SUBNET MASK serves to differentiate between the network ID and the host ID in the network. Input: four numerical values separated by points. Ask your network specialist for the values, e.g. 255.255.0.0
BROADCAST	The broadcast address of the control is required only if it differs from the standard setting. The standard setting is formed from the network ID and the host ID, for which all bits are set to 1, e.g. 160.1.255.255
ROUTER	Internet address of your default router. Enter the Internet address only if your network consists of several parts. Input: four numerical values separated by points. Ask your network specialist for the values, e.g. 160.1.0.2
HOST	Name under which the TNC identifies itself in the network
DOMAIN	Domain name of the control (is not evaluated until later)
NAMESERVER	Network address of the domain server (is not evaluated until later)

> You do not need to indicate the protocol with the iTNC 530. It uses the transmission protocol according to RFC 894.

Network settings specific to the device

- Press the soft key DEFINE MOUNT to enter the network settings for a specific device. You can define any number of network settings, but you can manage only seven at one time.



Setting	Meaning
MOUNTDEVICE	■ Connection via NFS: Name of the directory that is to be logged on. This is formed by the network address of the server, a colon and the name of the directory to be mounted. Input: four numerical values separated by points. Ask your network specialist for the values, e.g. 160.1.13.4. 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. ■ Connection to individual Windows computer: Enter the network name and the share name of the computer, e.g. //PC1791NT/C
MOUNTPOINT	Name that the TNC shows in the file manager for a connected device. Remember that the name must end with a colon.
FILESYSTEM-TYPE	File system type. **nfs**: Network File System **smb**: Windows network
OPTIONS for FILESYSTEM-TYPE=nfs	Data without spaces, separated by commas, and written in sequence. Switch between upper and lower case letters. **rsize=:** Packet size in bytes for data reception. Input range: 512 to 8192 **wsize=:** Packet size in bytes for data transmission. Input range: 512 to 8192 **time0=:** Time, in tenths of a second, after which the TNC repeats a Remote Procedure Call. Input range: 0 to 100 000. If there is no entry, the standard value 7 is used. Use higher values only if the TNC must communicate with the server through several routers. Ask your network specialist for the proper value. **soft=:** Definition of whether the TNC should repeat the Remote Procedure Call until the NFS server answers. "soft" entered: Do not repeat the Remote Procedure Call. "soft" not entered: Always repeat the Remote Procedure Call.

Setting	Meaning
OPTIONS for FILESYSTEM-TYPE=smb for direct connection to Windows networks	Data without spaces, separated by commas, and written in sequence. Switch between upper and lower case letters. **ip=:** ip address of PC to which the TNC is to be connected **username=:** User name under which the TNC is to log on **workgroup=:** Work group under which the TNC is to log on **password=:** Password that the TNC is to use for logon (up to 80 characters)
AM	Definition of whether the TNC upon switch-on should automatically connect with the network drive. 0: Do not automatically connect 1: Connect automatically

The entries **username**, **workgroup** and **password** in the OPTIONS column may not be necessary in Windows 95 and Windows 98 networks.

With the ENCODE PASSWORD soft key, you can encode the password defined under OPTIONS.

Defining a network identification

- Press the soft key DEFINE UID / GID to enter the network identification.

Setting	Meaning
TNC USER ID	Definition of the User Identification under which the end user accesses files in the network. Ask your network specialist for the proper value.
OEM USER ID	Definition of the User Identification under which the machine manufacturer accesses files in the network. Ask your network specialist for the proper value.
TNC GROUP ID	Definition of the group identification with which you access files in the network. Ask your network specialist for the proper value. The group identification is the same for end users and machine manufacturers.
UID for mount	Defines the user identification (UID) for the log-on procedure. **USER:** The user logs on with the USER identification. **ROOT:** The user logs on with the ID of the ROOT user, value = 0.

12.6 Configuring PGM MGT

Function

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

Note: see "Standard File Management," page 71, and see "Advanced File Management," page 78.

Changing the setting

- Select the file manager in the Programming and Editing mode of operation: press the PGM MGT key
- Press the MOD key to select the MOD function.
- 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.

12.7 Machine-Specific User Parameters

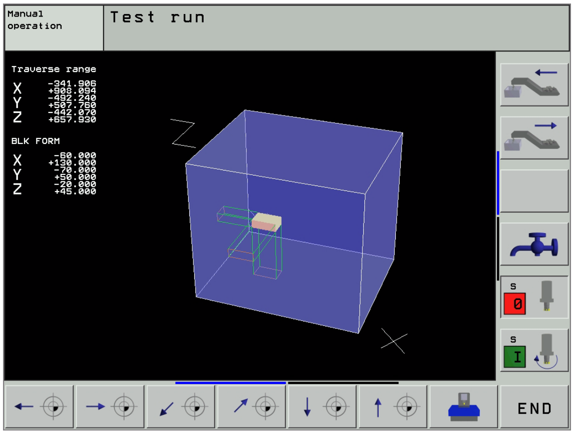
Function

To enable you to set machine-specific functions, your machine tool builder can define up to 16 machine parameters as user parameters.

This function is not available on all TNCs. Refer to your machine manual.

12.8 Showing the Workpiece in the Working Space

Function



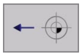
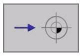
This MOD function enables you to graphically check the position of the workpiece blank in the machine's working space and to activate work space monitoring in the Test Run mode of operation. This function is activated with the BLANK IN WORK SPACE soft key.

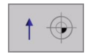
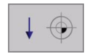
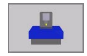
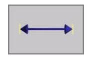
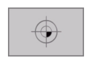
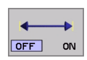
The TNC displays a cuboid for the working space. Its dimensions are shown in the "Traverse range" window. The TNC takes the dimensions for the working space from the machine parameters for the active traverse range. Since the traverse range is defined in the reference system of the machine, the datum 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 represents the blank form. The TNC takes its dimensions 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.

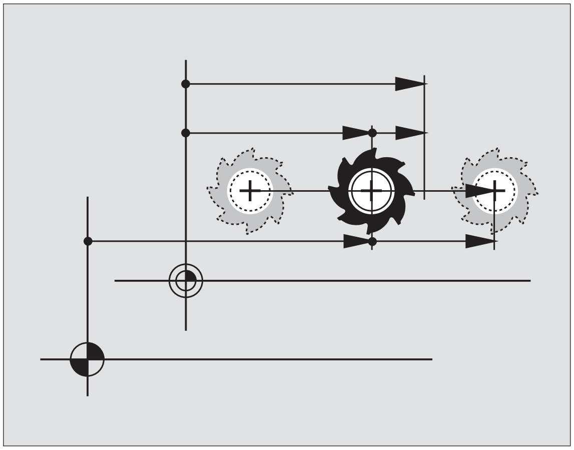
You can also activate the working-space monitor for the Test Run mode in order to test the program with the current datum and the active traverse ranges (see table below, last line).

Function	Soft key
Move workpiece blank to the left	
Move workpiece blank to the right	
Move workpiece blank forward	
Move workpiece blank backward	

Function	Soft key
Move workpiece blank upward	
Move workpiece blank downward	
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	
Show a position determined by the machine tool builder (e.g. tool change position) in the working space	
Show the workpiece datum in the working space	
Enable (ON) or disable (OFF) working-space monitoring	

12.9 Position Display Types

Function



In the Manual Operation mode and in the Program Run modes of operation, you can select the type of coordinates to be displayed.

The figure at right shows the different tool positions:

- Starting position
- Target position of the tool
- Workpiece datum
- Machine datum

The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the value presently commanded by the TNC	**NOML.**
Actual position; current tool position	**ACTL.**
Reference position; the actual position relative to the machine datum	**REF**
Distance remaining to the programmed position; difference between actual and target positions	**DIST.**
Servo lag: difference between nominal and actual positions (following error)	**LAG**
Deflection of the measuring touch probe	**DEFL.**
Traverses that were carried out with handwheel superpositioning (M118) (only Position display 2)	**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.

12.10 Unit of Measurement

Function

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g. X = 15.789 mm) set the Change mm/inches function to mm. The value is displayed to 3 decimal places.
- To select the inch system (e.g. X = 0.6216 inches) set the Change mm/inches function to inches. The value is displayed to 4 decimal places.

If you activate inch display, the TNC shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.

12.11 Select the Programming Language for $MDI

Function

The Program input MOD function lets you decide whether to program the $MDI file in HEIDENHAIN conversational dialog or in ISO format.

- To program the $MDI.H file in conversational dialog, set the Program input function to HEIDENHAIN
- To program the $MDI.I file according to ISO, set the Program input function to ISO

12.12 Selecting the Axes for Generating L Blocks

Function

> This function is only available with conversational dialog programming.

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 %11111Transfer the X, Y, Z, IV and V axes

Axis selection %01111Transfer the X, Y, Z and IV axes

Axis selection %00111Transfer the X, Y and Z axes

Axis selection %00011Transfer the X and Y axes

Axis selection %00001Transfer the X axis

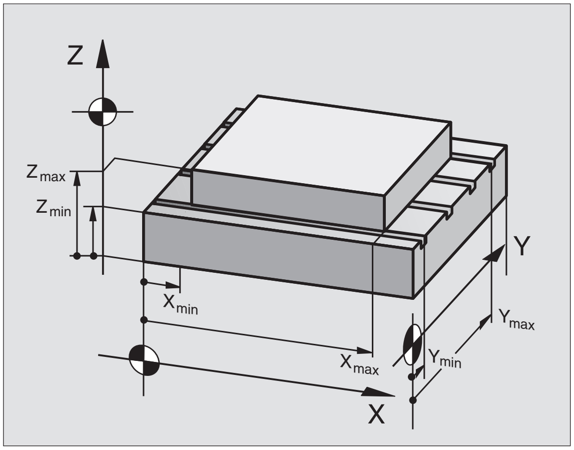
12.13 Enter the Axis Traverse Limits, Datum Display

Function

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

Possible application: Protecting an indexing fixture against tool collision.

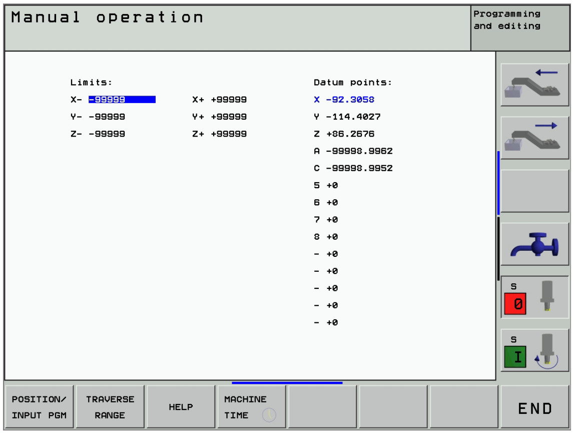
The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited through the TRAVERSE RANGE MOD function. With this function, you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum. If several traverse ranges are possible on your machine, you can set the limits for each range separately using the soft keys TRAVERSE RANGE (1) to TRAVERSE RANGE (3).



Working without additional traverse limits

To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 99 999 mm) as the TRAVERSE RANGE.

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 functions, 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.

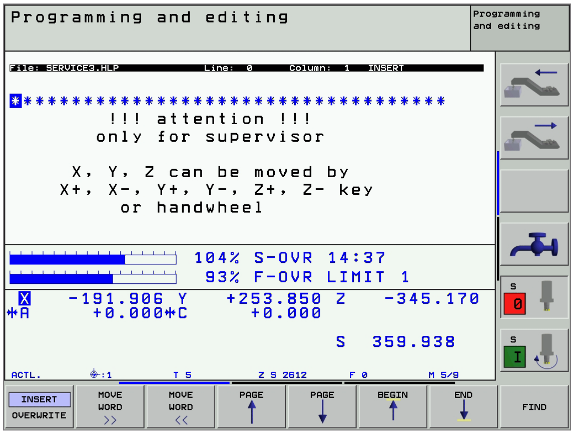
Datum display

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.

12.14 Displaying HELP Files

Function

Help files 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 in a help file. The figure at right shows the screen display of a help file.



> HELP files are not provided on every machine. Your machine tool builder can provide you with further information on this feature.

Selecting HELP files

- Press the MOD key to select the MOD function.

HELP

- To select the last active HELP file, press the HELP soft key.
- Call the file manager (PGM MGT key) and select a different help file, if necessary.

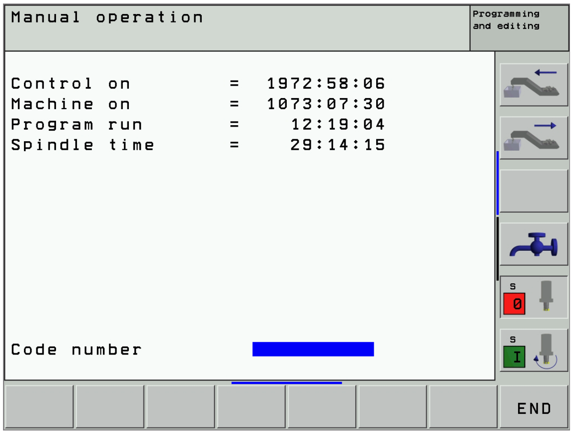
12.15 Display operating times

Function

The machine tool builder can provide further operating time displays. The machine tool manual provides further information.

The MACHINE TIME soft key enables you to show different operating time displays:

Operating time	Meaning
Control ON	Operating time of the control since commissioning
Machine ON	Operating time of the machine tool since commissioning
Program Run	Duration of controlled operation since commissioning



12.16 External Access

Function

The machine tool builder can configure teleservice settings with the LSV-2 interface. The machine tool manual provides further information.

The soft key SERVICE can be used to grant or restrict access through the LSV-2 interface.

With an entry in the configuration file TNC.SYS you can protect a directory and its subdirectories with a password. The password is requested when data from this directory is accessed from the LSV-2 interface. Enter the path and password for external access in the configuration file TNC.SYS.

The TNC.SYS file must be stored in the root directory TNC:\.

If you only supply one entry for the password, then the entire drive TNC:\ is protected.

You should use the updated versions of the HEIDENHAIN software TNCremo or TNCremoNT to transfer the data.

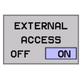
Entries in TNC.SYS	Meaning
REMOTE.TNCPASSWORD=	Password for LSV-2 access
REMOTE.TNCPRIVATEPATH=	Path to be protected

Example of TNC.SYS

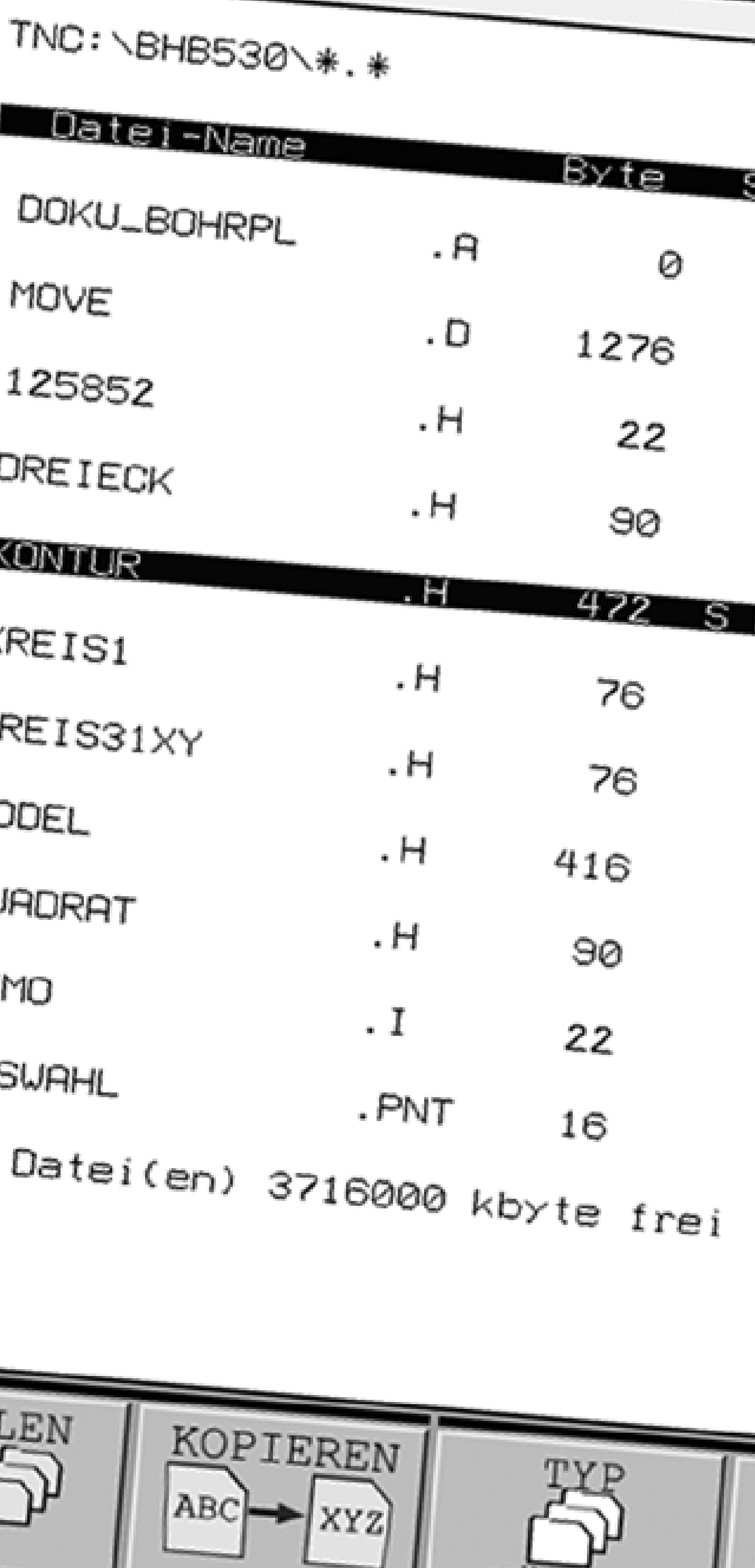
```
REMOTE.TNCPASSWORD=KR1402
REMOTE.TNCPRIVATEPATH=TNC:\RK
```

Permitting/Restricting external access

- Select any machine mode of operation.
- To select the MOD function, press the MOD key.



- Permit a connection to the TNC: Set the EXTERNAL ACCESS soft key to ON. The TNC will then permit data access through the LSV-2 interface. The password is requested when a directory that was entered in the configuration file TNC.SYS is accessed.
- Block connections to the TNC: Set the EXTERNAL ACCESS soft key to OFF. The TNC will then block access through the LSV-2 interface.



13

Tables and Overviews

13.1 General User Parameters

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

Some examples of user parameters are:

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

Input possibilities for machine parameters

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 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** 11/2 stop bits: **+0** 2 stop bit: **+64** 1 stop bit: **+128** 1 stop bit: **+192** 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**
Interface type for EXT1 (5030.0) and EXT2 (5030.1)	**MP5030.x** Standard transmission: **0** Interface for blockwise transfer: **1**

3-D touch probes and digitizing	
Select signal transmission	**MP6010** Touch probe with cable transmission: **0** Touch probe with infrared transmission: **1**
Probing feed rate for triggering touch probes	**MP6120** **1** 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]

3-D touch probes and digitizing	
Measure center misalignment of the stylus when calibrating a triggering touch probe	**MP6160** No 180° rotation of the 3-D touch probe during calibration: **0** M function for 180° rotation of the touch probe during calibration: **1** to **999**
M function for orienting the infrared sensor before each measuring cycle	**MP6161** Function inactive: **0** Orientation directly through the NC: **-1** M function for orienting the touch probe: **1** to **999**
Angle of orientation for the infrared sensor	**MP6162** **0** to **359.9999** [°]
Difference between the current angle of orientation and the angle of orientation set in MP 6162; when the entered difference is reached, an oriented spindle stop is to be carried out.	**MP6163** **0** to **3.0000** [°]
Automatically orient the infrared sensor before probing to the programmed probing direction	**MP6165** Function inactive: **0** Orient infrared sensor: **1**
Multiple measurement for programmable probe function	**MP6170** **1** to **3**
Confidence range for multiple measurement	**MP6171** **0.001** to **0.999** [mm]
Automatic calibration cycle: Center of the calibration ring in the X axis referenced to the machine datum	**MP6180.0 (traverse range 1) to MP6180.2 (traverse range3)** **0** to **99 999.9999** [mm]
Automatic calibration cycle: Center of the calibration ring in the Y axis referenced to the machine datum	**MP6181.0 (traverse range 1) to MP6181.2 (traverse range 3)** **0** to **99 999.9999** [mm]
Automatic calibration cycle: Upper edge of the calibration ring in the Z axis referenced to the machine datum	**MP6182.0 (traverse range 1) to MP6182.2 (traverse range 3)** **0** to **99 999.9999** [mm]
Automatic calibration cycle: Distance below the upper edge of the ring where the calibration is carried out by the TNC	**MP6185.0 (traverse range 1) to MP6185.2 (traverse range 3)** **0.1** to **99 999.9999** [mm]
Radius measurement with the TT 130 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 130, with constant tolerance: **+0** Calculate feed rate for second measurement with TT 130, with variable tolerance: **+1** Constant feed rate for second measurement with TT 130: **+2**

3-D touch probes and digitizing	
Maximum permissible measuring error with TT 130 during measurement with rotating tool Required for calculating the probing feed rate in connection with MP6570	**MP6510.0** **0.001** to **0.999** [mm] (recommended input value: 0.005 mm) **MP6510.1** **0.001** to **0.999** [mm] (recommended input value: 0.01 mm)
Feed rate for probing a stationary tool with the TT 130	**MP6520** **1** to **3000** [mm/min]
Radius measurement with the TT 130: Distance from lower edge of tool to upper edge of stylus	**MP6530.0 (traverse range 1) to MP6530.2 (traverse range 3)** **0.001** to **99.9999** [mm]
Set-up clearance in the tool axis above the stylus of the TT 130 for pre-positioning	**MP6540.0** **0.001** to **30 000.000** [mm]
Clearance zone in the machining plane around the stylus of the TT 130 for pre-positioning	**MP6540.1** **0.001** to **30 000.000** [mm]
Rapid traverse for TT 130 in the probe cycle	**MP6550** **10** to **10 000** [mm/min]
M function for spindle orientation when measuring individual teeth	**MP6560** **0** to **999**
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]
Measuring rotating tools: Permissible rotational rpm	**MP6572** **0.000** to **1000.000** [rpm] If you enter 0, the speed is limited to 1000 rpm

3-D touch probes and digitizing	
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)** X axis **MP6581.1 (traverse range 2)** Y axis **MP6581.2 (traverse range 2)** Z axis **MP6582.0 (traverse range 3)** X axis **MP6582.1 (traverse range 3)** Y axis **MP6582.2 (traverse range 3)** Z axis
Monitoring the position of rotary axes and parallel axes	**MP6585** Function inactive: **0** Function active: **1**
Defining the rotary axes and parallel axes to be monitored	**MP6586.0** Do not monitor the position of the A axis: **0** Monitor the position of the A axis: **1** **MP6586.1** Do not monitor the position of the B axis: **0** Monitor the position of the B axis: **1** **MP6586.2** Do not monitor the position of the C axis: **0** Monitor the position of the C axis: **1** **MP6586.3** Do not monitor the position of the U axis: **0** Monitor the position of the U axis: **1** **MP6586.4** Do not monitor the position of the V axis: **0** Monitor the position of the V axis: **1** **MP6586.5** Do not monitor the position of the W axis: **0** Monitor the position of the W axis: **1**

TNC displays, TNC editor	
Cycles 17, 18 and 207: Oriented spindle stop at beginning of cycle	**MP7160** Oriented spindle stop: **0** No oriented spindle stop: **1** Bit 1 to bit 3: Function
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**
Disabling the selection of file types	**MP7224.0** All file types selectable via soft key: **+0** Disable selection of HEIDENHAIN programs (soft key SHOW .H): **+1** Disable selection of ISO programs (soft key SHOW .I): **+2** Disable selection of tool tables (soft key SHOW .T): **+4** Disable selection of datum tables (soft key SHOW .D): **+8** Disable selection of pallet tables (soft key SHOW .P): **+16** Disable selection of text files (soft key SHOW .A):**+32** Disable selection of point tables (soft key SHOW .PNT): **+64**
Disabling the editor for certain file types **Note:** If a particular file type is inhibited, the TNC will erase all files of this type.	**MP7224.1** Do not disable editor: **+0** Disable editor for ■ HEIDENHAIN programs: **+1** ■ ISO programs: **+2** ■ Tool tables: **+4** ■ Datum tables: **+8** ■ Pallet tables: **+16** ■ Text files: **+32** ■ Point tables: **+64**
Configure pallet files	**MP7226.0** Pallet table inactive: **0** Number of pallets per pallet table: **1** to **255**
Configure datum files	**MP7226.1** Datum table inactive: **0** Number of datums per datum table: **1** to **255**
Program length for program check	**MP7229.0** Blocks **100** to **9999**
Program length up to which FK blocks are permitted	**MP7229.1** Blocks **100** to **9999**

TNC displays, TNC editor	
Dialog language	**MP7230.0 to MP7230.3** English: **0** German: **1** Czech: **2** French: **3** Italian: **4** Spanish: **5** Portuguese: **6** Swedish: **7** Danish: **8** Finnish: **9** Dutch: **10** Polish: **11** Hungarian: **12** Reserved: **13** Russian: **14**
Internal clock of the TNC	**MP7235** Universal time (Greenwich time): **0** Central European Time (CET): **1** Central European Summer Time: **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 **30 000**
Configure pocket tables	**MP7261.0 (magazine 1)** **MP7261.1 (magazine 2)** **MP7261.2 (magazine 3)** **MP7261.3 (magazine 4)** Inactive: **0** Number of pockets in the tool magazine: **1** to **254** If the value 0 is entered in MP7261.1 to MP7261.3, only one tool magazine will be used.
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**
POCKET TABLE soft key	**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**

TNC displays, TNC editor	
Configure tool table (enter 0 to omit from table); Column number in the tool table	**MP7266.0** Tool name – NAME: **0** to **32**; column width: 16 characters **MP7266.1** Tool length – L: **0** to **32**; column width: 11 characters **MP7266.2** Tool radius – R: **0** to **32**; column width: 11 characters **MP7266.3** Tool radius 2 – R2: **0** to **32**; column width: 11 characters **MP7266.4** Oversize length – DL: **0** to **32**; column width: 8 characters **MP7266.5** Oversize radius – DR: **0** to **32**; column width: 8 characters **MP7266.6** Oversize radius 2 – DR2: **0** to **32**; column width: 8 characters **MP7266.7** Tool locked – TL: **0** to **32**; column width: 2 characters **MP7266.8** Replacement tool – RT: **0** to **32**; column width: 3 characters **MP7266.9** Maximum tool life – TIME1: **0** to **32**; column width: 5 characters **MP7266.10** Maximum tool life for TOOL CALL – TIME2: **0** to **32**; column width: 5 characters **MP7266.11** Current tool life – CUR. TIME: **0** to **32**; column width: 8 characters **MP7266.12** Tool comment – DOC: **0** to **32**; column width: 16 characters **MP7266.13** Number of teeth – CUT.: **0** to **32**; column width: 4 characters **MP7266.14** Tolerance for wear detection in tool length – LTOL: **0** to **32**; column width: 6 characters **MP7266.15** Tolerance for wear detection in tool radius – RTOL: **0** to **32**; column width: 6 characters **MP7266.16** Cutting direction – DIRECT.: **0** to **32**; column width: 7 characters **MP7266.17** PLC status – PLC: **0** to **32**; column width: 9 characters **MP7266.18** Offset of the tool in the tool axis in addition to MP6530 – TT:L-OFFS: **0** to **32** column width: 11 characters **MP7266.19** Offset of the tool between stylus center and tool center – TT:R-OFFS: **0** to **32** column width: 11 characters **MP7266.20** Tolerance for break detection in tool length – LBREAK: **0** to **32**; column width: 6 characters **MP7266.21** Tolerance for break detection in tool radius – RBREAK: **0** to **32**; column width: 6 characters **MP7266.22** Tooth length (Cycle 22) – LCUTS: **0** to **32**; column width: 11 characters **MP7266.23** Maximum plunge angle (Cycle 22) – ANGLE.: **0** to **32**; column width: 7 characters **MP7266.24** Tool type –TYP: **0** to **32**; column width: 5 characters **MP7266.25** Tool material – TMAT: **0** to **32**; column width: 16 characters **MP7266.26** Cutting data table – CDT: **0** to **32**; column width: 16 characters

TNC displays, TNC editor	
Configure tool table (enter 0 to omit from table); Column number in the tool table	**MP7266.27** PLC value – PLC-VAL: **0** to **32**; column width: 11 characters **MP7266.28** Center misalignment in reference axis – CAL-OFF1: **0** to **32**; column width: 11 characters **MP7266.29** Center misalignment in minor axis – CAL-OFF2: **0** to **32**; column width: 11 characters **MP7266.30** Spindle angle for calibration – CALL-ANG: **0** to **32**; column width: 11 characters **MP7266.31** Tool type for the pocket table–PTYP: **0** to **32**; column width: 2 characters
Configure pocket table; Column number in the pocket table (enter 0 to omit from table)	**MP7267.0** Tool number – T: **0** to **18** **MP7267.1** Special tool – ST: **0** to **18** **MP7267.2** Fixed pocket – F: **0** to **18** **MP7267.3** Pocket locked – L: **0** to **18** **MP7267.4** PLC status – PLC: **0** to **18** **MP7267.5** Tool name from tool table – TNAME: **0** to **18** **MP7267.6** Comment from tool table – DOC: **0** to **18**
Configure pocket table; Column number in the pocket table when using a box magazine (enter 0 to omit from table)	**MP7267.7 to MP7267.17** Evaluated by the PLC: **0** to **18**
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**
Decimal character	**MP7280** The decimal character is a comma: **0** The decimal character is a point: **1**
Display mode	**MP7281.0 Programming and Editing operating mode** **MP7281.1 Program Run operating modes** Always display multiple line blocks completely: **0** Display multiline blocks completely if the multiline block is the active block: **1** Display multiline blocks completely if the multiline block is being edited: **2**
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**

TNC displays, TNC editor	
Display step for the spindle position	**MP7289** 0,1 °: **0** 0,05 °: **1** 0,01 °: **2** 0,005 °: **3** 0,001 °: **4** 0,0005 °: **5** 0,0001 °: **6**
Display step	**MP7290.0 (X axis) to MP7290.8 (9th axis)** 0.1 mm: **0** 0.05 mm: **1** 0.01 mm: **2** 0.005 mm: **3** 0.001 mm: **4** 0.0005 mm: **5** 0.0001 mm: **6**
Disable datum setting	**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 IVth axis: **+8** Disable datum setting in the Vth 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 disable datum setting: **0** Disable datum setting with the orange axis keys: **1**
Reset status display, Q parameters and tool data	**MP7300** Reset all when a program is selected: **0** Reset 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: **+1** Projection in three planes according to ISO 6433, projection method 2: **+1** Do not rotate coordinate for graphic display: **+0** Rotate coordinate system for graphic display by 90°: **+2** Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the old datum: **+0** Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the new datum: **+4** Do not show cursor position during projection in three planes: **+0** Show cursor position during projection in three planes: **+8**

TNC displays, TNC editor	
Graphic simulation without programmed tool axis: Tool radius	**MP7315** **0** to **99 999.9999** [mm]
Graphic simulation without programmed tool axis: Penetration depth	**MP7316** **0** to **99 999.9999** [mm]
Graphic simulation without programmed tool axis: M function for start	**MP7317.0** **0** to **88** (0: Function inactive)
Graphic simulation without programmed spindle axis: M function for end	**MP7317.1** **0** to **88** (0: Function inactive)
Screen saver Enter the time after which the TNC should start the screen saver	**MP7392** **0** to **99** [min] (0: Function inactive)

Machining and program run	
Effect of Cycle 11 SCALING FACTOR	**MP7410** SCALING FACTOR effective in 3 axes: **0** SCALING FACTOR effective in the working plane only: **1**
Manage tool data/calibration data	**MP7411** Overwrite current tool data by the calibrated data from the 3-D touch probe system: **+0** Current tool data are retained: **+1** Manage calibrated data in the calibration menu: **+0** Manage calibrated data in the tool table: **+2**

Machining and program run	
SL Cycles	**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**
Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET MILLING: Overlap factor	**MP7430** **0.1** to **1.414**
Permissible deviation of circle radius between circle end point and circle starting point	**MP7431** **0.0001** to **0.016** [mm]
Operation of various miscellaneous functions M **Note:** 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** k_V factors switchable through M105 and M106: **+8** 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 for positioning with rotary axes Not active: **+0** Exact stop for positioning with rotary axes Active: **+64**
Error message during cycle call	**MP7441** Error message when M3/M4 not active: **0** Suppress error message when M3/M4 not active: **+1** Reserved: **+2** Suppress error message when positive depth programmed: **+0** Output error message when negative depth programmed: **+4**
M function for spindle orientation in the fixed cycles	**MP7442** Function inactive: **0** Orientation directly through the NC: **-1** M function for orienting the spindle: **1** to **999**

Machining and program run	
Maximum contouring speed at feed rate override setting of 100% in the Program Run modes	**MP7470** **0** to **99 999** [mm/min]
Feed rate for rotary-axis compensation movements	**MP7471** **0** to **99 999** [mm/min]
NC software 340 420-03 and previous software versions: Datums from datum table are referenced to the **NC software 340 420-03 and later: No function**	**MP7475** Workpiece datum: **0** Machine datum: **1**

13.2 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices

The interface complies with the requirements of EN 50 178 for “low voltage electrical separation.”

When using the 25-pin adapter block:

TNC		Adapter block 310 085-01			Connecting cable 365 725-xx				
Male	**Assignment**	**Female**	**Color**	**Female**	**Male**	**Female**	**Male**	**Color**	**Female**
1	Do not assign	1		1	1	1	1	WH/BN	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

When using the 9-pin adapter block:

TNC		Connecting cable 355 484-xx			Adapter block 363 987-02		Connecting cable 366 964-xx		
Male	**Assignment**	**Female**	**Color**	**Male**	**Female**	**Male**	**Female**	**Color**	**Female**
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	WH/GN	8	8	8	8	WH/GN	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

Non-HEIDENHAIN devices

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

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

Adapter block 363 987-02		Connecting cable 366 964-xx		
Female	**Male**	**Female**	**Color**	**Female**
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	WH/GN	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

RS-422/V.11 interface

Only non-HEIDENHAIN devices are connected to the RS-422 interface.

> The interface complies with the requirements of EN 50 178 for “low voltage electrical separation.”
>
> The pin layouts on the TNC logic unit (X28) and on the adapter block are identical.

TNC		Connecting cable 355 484-xx			Adapter block 363 987-01	
Female	**Assignment**	**Male**	**Color**	**Female**	**Male**	**Female**
1	RTS	1	Red	1	1	1
2	DTR	2	Yellow	2	2	2
3	$\overline{\text{RXD}}$	3	White	3	3	3
4	$\overline{\text{TXD}}$	4	Brown	4	4	4
5	Signal GND	5	Black	5	5	5
6	CTS	6	Violet	6	6	6
7	DSR	7	Gray	7	7	7
8	RXD	8	WH/GN	8	8	8
9	TXD	9	Green	9	9	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.

Ethernet interface RJ45 socket

Maximum cable length: Unshielded: 100 m
Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	

13.3 Technical Information

Explanation of symbols

■ Standard
● Axis option
❍ Software option 1
□ Software option 2

User functions	
Description	■ Basic version: 3 axes plus spindle ● 4. NC axis plus auxiliary axis or ● 8 additional axes or 7 additional axes plus 2nd spindle ■ Digital current and speed control
Programming	HEIDENHAIN conversational and ISO formats
Position entry	■ Nominal positions for line segments and arcs in Cartesian or polar coordinates ■ Absolute or incremental dimensions ■ Display and entry in mm or inches ■ Display of the handwheel path during machining with handwheel superimposition
Tool compensation	■ Tool radius in the working plane and tool length ■ Calculating the radius-compensated contour up to 99 blocks in advance (M120) ■ Three-dimensional tool-radius compensation for subsequent changing of tool data without having to recalculate the program
Tool tables	Multiple tool tables with any number of tools
Cutting data tables	Cutting data tables for automatic calculation of spindle speed and feed rate from tool-specific data (cutting speed, feed per tooth)
Constant cutting speed	■ With respect to the path of the tool center ■ With respect to the cutting edge
Background programming	Create one program with graphical support while another program is running.
3-D machining (software option 2)	□ Motion control with minimum jerk □ 3-D compensation through surface normal vectors □ Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = **T**ool **C**enter **P**oint **M**anagement) □ Keeping the tool normal to the contour □ Tool radius compensation normal to the direction of traverse and the tool direction □ Spline interpolation
Rotary table machining (software option 1)	❍ Programming of cylindrical contours as if in two axes ❍ Feed rate in length per minute

User functions	
Contour elements	■ Straight line ■ Chamfer ■ Circular path ■ Circle center ■ Circle radius ■ Tangentially connecting circle ■ Corner rounding
Contour approach and departure	■ Via straight line: tangential or perpendicular ■ Via circular arc
FK free contour programming	■ FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	■ Subprograms ■ Program section repeat ■ Program as subprogram
Fixed cycles	■ Drilling cycles for drilling, pecking, reaming, boring, tapping with a floating tap holder, rigid tapping ■ Cycles for milling internal and external threads ■ Milling and finishing rectangular and circular pockets ■ Cycles for multipass milling of flat and twisted surfaces ■ Cycles for milling linear and circular slots ■ Linear and circular hole patterns ■ Contour pockets—also with contour-parallel machining ■ Contour train ■ OEM cycles (special cycles developed by the machine tool builder) can also be integrated
Coordinate transformation	■ Datum shift, rotation, mirroring ■ Axis-specific scaling ❍ Tilting the working plane (software option 1)
Q parameters Programming with variables	■ Mathematic functions =, +, –, *, /, sin α , cos α , angle α of sin α and cos α , $\sqrt{a^2 + b^2}$ $\sqrt{a}$ ■ Logical comparisons (=, =/, <, >) ■ Calculating with parentheses ■ tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, the constant π , negation, truncation of digits before or after the decimal point ■ Functions for calculating circles
Programming support	■ Pocket calculator ■ Context-sensitive help function for error messages ■ Graphical support during programming of cycles ■ Comment blocks in the NC program
Actual position capture	■ Actual positions can be transferred directly into the NC program

User functions	
Test Run graphics Display modes	Graphic simulation before a program run, even while another program is being run ■ Plan view / projection in 3 planes / 3-D view ■ Magnification of details
Interactive Programming graphics	■ In the Programming and Editing mode, the contour of the NC blocks is drawn on screen while they are being entered (2-D pencil-trace graphics), even while another program is running
Program Run graphics Display modes	■ Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■ Calculating the machining time in the Test Run mode of operation ■ Display of the current machining time in the Program Run modes
Returning to the contour	■ Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining ■ Program interruption, contour departure and reapproach
Datum tables	■ Several datum tables
Pallet tables	■ Pallet tables (with as many entries as desired for the selection of pallets, NC programs and datums) can be machined workpiece by workpiece or tool by tool
Touch Probe Cycles	■ Calibrating a touch probe ■ Compensation of workpiece misalignment, manual or automatic ■ Datum setting, manual or automatic ■ Automatic workpiece measurement ■ Cycles for automatic tool measurement

Specifications	
Components	■ MC 422 main computer ■ CC 422 controller unit ■ Keyboard ■ TFT 10.4-inch or 15.1-inch flat-panel display with soft keys
Program memory	■ Hard disk with at least 2 GB for NC programs
Input resolution and display step	■ To 0.1 µm for linear axes ■ To 0.0001° for angular axes
Input range	■ Maximum 99 999.999 mm (3937 in.) or 99 999.999°
Interpolation	■ Line in 4 axes □ Line in 5 axes (subject to export permit) (software option 1) ■ Arc in 2 axes ○ Arc in 3 axes with tilted working plane (software option 1) ■ Helix: Combination of circular and linear motion ■ Spline: Execution of splines (3rd degree polynomials)

Specifications	
Block processing time 3-D straight line without radius compensation	■ 3.6 ms □ 0.5 ms (software option 2)
Axis feedback control	■ Position loop resolution: Signal period of the position encoder/1024 ■ Cycle time of position controller: 1.8 ms ■ Cycle time of speed controller: 600 µs ■ Cycle time of current controller: minimum 100 µs
Traverse range	■ Maximum 100 m (3973 inches)
Spindle speed	■ Maximum 40 000 rpm (with 2 pole pairs)
Error compensation	■ Linear and nonlinear axis error, backlash, reversal spikes during circular movements, thermal expansion ■ Stick-slip friction
Data interfaces	■ One each RS-232-C /V.24 and RS-422 / V.11 max. 115 kilobaud ■ Expanded data interface with LSV-2 protocol for remote operation of the TNC through the data interface with the HEIDENHAIN software TNCremo ■ Ethernet interface 100 Base T approx. 2 to 5 megabaud (depending on file type and network load)
Ambient temperature	■ Operation: 0 °C to +45 °C (32 °F to 113 °F) ■ Storage: –30 °C to +70 °C (–22 °F to 158 °F)

Accessories	
Electronic handwheels	■ One **HR 410:** portable handwheel or ■ One **HR 130:** panel-mounted handwheel or ■ Up to three **HR 150:** panel-mounted handwheels via HRA 110 handwheel adapter
Touch probes	■ **TS 220:** 3-D touch trigger probe with cable connection, or ■ **TS 632:** 3-D touch trigger probe with infrared transmission ■ **TT 130:** 3-D touch trigger probe for workpiece measurement

Software option 1	
Rotary table machining	❍ Programming of cylindrical contours as if in two axes ❍ Feed rate in length per minute
Coordinate transformations	❍ Tilting the working plane
Interpolation	❍ Circle in 3 axes (with tilted working plane)

Software option 2	
3-D machining	☐ Motion control with minimum jerk ☐ 3-D compensation through surface normal vectors ☐ Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = **T**ool **C**enter **P**oint **M**anagement) ☐ Keeping the tool normal to the contour ☐ Tool radius compensation normal to the direction of traverse and the tool direction ☐ Spline interpolation
Interpolation	☐ Line in 5 axes (subject to export permit)
Block processing time	☐ 0.5 ms

Input format and unit of TNC functions	
Positions, coordinates, circle radii, chamfer lengths	–99 999.9999 to +99 999.9999 (5.4: places before decimal point, places after decimal point) [mm]
Tool numbers	0 999 99 to 32 767.9 999 99 (5.1)
Tool names	16 characters, enclosed by quotation marks with TOOL CALL. Permitted special characters: #, $, %, &, -
Delta values for tool compensation	–99.9999 to +99.9999 (2.4) [mm]
Spindle speeds	0 to 99 999.999 (5.3) [rpm]
Feed rates	0 to 99 999.999 (5.3) [mm/min] or [mm/rev]
Dwell time in Cycle 9	0 to 3600.000 (4.3) [s]
Thread pitch in various cycles	–99.9999 to +99.9999 (2.4) [mm]
Angle of spindle orientation	0 to 360.0000 (3.4) [°]
Angle for polar coordinates, rotation, tilting the working plane	-360.0000 to 360.0000 (3.4) [°]
Polar coordinate angle for helical interpolation (CP)	–5400.0000 to +5400.0000 (4.4) [°]
Datum numbers in Cycle 7	0 999 99 to 2 999 999 99 (4.0)
Scaling factor in Cycles 11 and 26	0.000 001 to 99.999 999 (2.6)
Miscellaneous functions M	0 999 99 to 999 999 99 (1.0)
Q parameter numbers	0 999 99 to 399 999 99 (1.0)
Q parameter values	-99 999.9999 999 99 to +99 999.9999 999 99 (5.4)
Labels (LBL) for program jumps	0 999 99 to 254 999 99 (3.0)
Number of program section repeats REP	1 999 99 to 65 534 999 99 (5.0)
Error number with Q parameter function FN14	0 999 99 to 1 099 999 99 (4.0)
Spline parameter K	–9.999 999 99 to +9.999 999 99 (1.8)
Exponent for spline parameter	-255 999 99 to 255 999 99 (3.0)
Surface-normal vectors N and T with 3-D compensation	–9.999 999 99 to +9.999 999 99 (1.8)

13.4 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:

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

Battery type: 1 Lithium battery, type CR 2450N (Renata)
ID Nr. 315 878-01

1. The backup battery is at the back of the MC 422.
2. Exchange the battery. The new battery can only be inserted correctly.

13.5 Addresses (ISO)

G functions

Group	G	Function	Blockwise function	Note
Positioning	00	Straight-line interpolation, Cartesian coordinates, rapid traverse		page 165
	01	Straight-line interpolation, Cartesian coordinates		page 165
	02	Circular interpolation, Cartesian coordinates, clockwise	■ (with R)	page 169
	03	Circular interpolation, Cartesian coordinates, counterclockwise	■ (with R)	page 169
	05	Circular interpolation, Cartesian coordinates, without indication of direction		page 169
	06	Circular interpolation, Cartesian coordinates, tangential contour approach		page 172
	07	Paraxial positioning block	■	
	10	Straight-line interpolation, polar coordinates, rapid traverse		page 178
	11	Straight-line interpolation, polar coordinates		page 178
	12	Circular interpolation, polar coordinates, clockwise		page 178
	13	Circular interpolation, polar coordinates, counterclockwise		page 178
	15	Circular interpolation, polar coordinates, without indication of direction		page 178
	16	Circular interpolation, polar coordinates, tangential contour approach		page 179
Machining contours, approaching/departing	24	Chamfer with length R		page 166
	25	Corner rounding with radius R		page 167
	26	Tangential approach of a contour with R		page 162
	27	Tangential departure of a contour with R		page 162
Cycles for drilling, tapping and thread milling	83	Pecking		page 224
	84	Tapping with a floating tap holder		page 239
	85	Rigid tapping		page 242
	86	Thread cutting		page 245
	200	Drilling		page 225
	201	Reaming		page 227
	202	Boring		page 229
	203	Universal drilling		page 231
	204	Back boring		page 233
	205	Universal pecking		page 235
	206	Tapping with a floating tap holder		page 240
	207	Rigid tapping		page 243
	208	Bore milling		page 237
	209	Tapping with chip breaking		page 246
	262	Thread milling		page 250
	263	Thread milling/countersinking		page 252
	264	Thread drilling/milling		page 255
	265	Helical thread drilling/milling		page 258
	267	Outside thread milling		page 261

Group	G	Function	Blockwise function	Note
Cycles for Milling Pockets, Studs and Slots	74 75 76 77 78 210 211 212 213 214 215	Slot milling Rectangular pocket milling in clockwise direction Circular path in counterclockwise direction Circular pocket milling in clockwise direction Circular pocket milling in counterclockwise direction Slot milling with reciprocating plunge Round slot with reciprocating plunge Rectangular pocket finishing Rectangular stud finishing Circular pocket finishing Circular stud finishing		page 283 page 271 page 271 page 277 page 277 page 285 page 288 page 273 page 275 page 279 page 281
Cycles for creating point patterns	220 221	Circular pattern Linear pattern		page 294 page 296
Cycles for creating complex contours	37 56 57 58 59 37 120 121 122 123 124 125 127 128	Definition of pocket contour Pilot drilling of the contour pocket (with G37) SLI Rough-out of the contour pocket (with G37) SLI Contour milling in clockwise direction (with G37) SLI Contour milling in counterclockwise direction (with G37) SLI Definition of pocket contour Contour data Pilot drilling (with G37) SLII Reaming (with G37) SLII Floor finishing (with G37) SLII Side finishing (with G37) SLII Contour train (with G37) Cylinder surface (with G37) Cylindrical surface slot (with G37)		page 302 page 303 page 304 page 305 page 305 page 306 page 311 page 312 page 313 page 314 page 315 page 316 page 318 page 320
Cycles for multipass milling	60 230 231	3-D data Multipass milling of plane surfaces Multipass milling of tilted surfaces		page 340 page 341 page 343
Coordinate transformation cycles	28 53 54 72 73 80	Mirror image Datum shift in a datum table Datum shift in program Scaling factor Rotation of the coordinate system Machining plane		page 355 page 350 page 349 page 358 page 357 page 359
Special Cycles	04 36 39 62	Dwell time Oriented spindle stop Cycle for program call, program call via G79 Tolerance deviation for fast contour milling	■ ■ 	page 366 page 368 page 367 page 369
Cycles for measurement of workpiece misalignment	400 401 402 403 404 405	Basic rotation from two points Basic rotation from two holes Basic rotation from two studs Compensating misalignment with rotary axis Setting a basic rotation directly Compensating misalignment with the C axis	■ ■ ■ ■ ■ ■	See User's Manual "Touch Probe Cycles"

Group	G	Function	Blockwise function	Note
Cycles for automatic datum setting	410 411 412 413 414 415 416 417 418	Datum in center of rectangular pocket Datum in center of rectangular stud Datum in center of circular pocket/hole Datum in center of circular stud Datum in inside corner Datum in outside corner Datum in center of bolt hole circle Datum in the touch probe axis Datum in intersection of two connecting lines each connecting two holes	■ ■ ■ ■ ■ ■ ■ ■ ■	See User's Manual "Touch Probe Cycles"
Cycles for automatic workpiece measurement	55 420 421 422 423 424 425 426 427 430 431	Measuring any coordinate in any axis Measuring angles Measuring position and diameter of a circular pocket/hole Measuring position and diameter of a circular stud Measuring position and diameter of a rectangular pocket Measuring position and diameter of a rectangular stud Measuring the slot width Measuring a ridge Measuring any coordinate in any axis Measuring position and diameter of a bolt hole circle Measuring a plane	■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■	See User's Manual "Touch Probe Cycles"
Cycles for automatic tool measurement	480 481 482 483	Calibrating the TT Measuring tool length Measuring tool radius Measuring tool length and radius	■ ■ ■ ■	See User's Manual "Touch Probe Cycles"
Cycles in general	79	Call the cycle	■	page 216
Selection of the machining plane	17 18 19 20	Plane selection XY, tool axis Z Plane selection ZX, tool axis Y Plane selection YZ, tool axis X Tool axis IV		page 140
Capture of coordinates	29	Transfer the last nominal position value as a pole		page 168
Define the workpiece blank	30 31	Define workpiece blank for graphics, min. point Define workpiece blank for graphics, max. point		page 91
Influencing the program run	38	Program run STOPP		
	40 41 42 43 44	No tool compensation (R0) Tool radius compensation, to the left of the contour (RL) Tool radius compensation, to the right of the contour (RR) Paraxial compensation, lengthening (R+) Paraxial compensation, shortening (R–)		page 144
Tools	51 99	Next tool number (in active central tool memory) Tool definition	■ ■	page 141 page 132
Unit of measure	70 71	Unit of measure: inches (set at start of program) Unit of measure: millimeters (set at start of program)		page 92

Group	G	Function	Blockwise function	Note
Dimensions	90 91	Absolute dimensions Incremental dimensions		page 67 page 67
Subprograms	98	Setting a label number	■	

Assigned addresses

Address	Function
%	Program start or program call
#	Datum number with Cycle G53
A **B** **C**	Rotation about X axis Rotation about Y axis Rotation about Z axis
D	Definition of parameters (program parameters Q)
DL **DR**	Length wear compensation with tool call Radius wear compensation with tool call
E	Tolerance for M112 and M124
F **F** **F** **F**	Feed rate Dwell time with G04 Scaling factor with G72 Factor for feed-rate reduction with M103
G	Preparatory function, cycle definition
H **H** **H**	Polar coordinates angle in incremental value/absolute value Rotation angle with G73 Tolerance angle for M112
I **J** **K**	Z coordinate of the circle center/pole Y coordinate of the circle center/pole Z coordinate of the circle center/pole
L **L** **L**	Setting a label number with G98 Jump to a label number Tool length with G99
LA	Number of blocks for block scan with M120
M	Miscellaneous Functions
N	Block number
P **P**	Cycle parameters in machining cycles Parameters in parameter definitions
Q	Program parameters/Cycle parameters

Address	Function
R R R R R	Polar coordinate radius Circular radius with G02/G03/G05 Rounding radius with G25/G26/G27 Chamfer section with G24 Tool radius with G99
S S	Spindle speed Oriented spindle stop with G36
T T	Tool definition with G99 Tool call
U V W	Linear movement parallel to X axis Linear movement parallel to Y axis Linear movement parallel to Z axis
X Y Z	X axis Y axis Z axis
*	End of block

Parameter functions

Parameter definition	Function	Note
D00	Assign	page 389
D01 D02 D03 D04	Addition Subtraction Multiplication Division	page 389 page 389 page 389 page 389
D05	Root	page 389
D06 D07	Sine Cosine	page 392 page 392
D08	Root sum of squares	page 392
D09 D10 D11 D12	If equal, go to If not equal, go to If greater than, go to If less than, go to	page 394 page 394 page 394 page 394
D13	Angle from c · sin a and c · cos a	page 392
D14	Error number	page 397
D15	Print	page 399
D19	Transfer of values to the PLC	page 399

Q

R

S

T

Table of Miscellaneous Functions

M	Effect	Effective at block start	end	Page
M00	Stop program/Spindle STOP/Coolant OFF		■	page 187
M01	Optional program STOP		■	page 436
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	page 187
M03	Spindle ON clockwise	■		page 187
M04	Spindle ON counterclockwise	■		
M05	Spindle STOP		■	
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP		■	page 187
M08	Coolant ON	■		page 187
M09	Coolant OFF		■	
M13	Spindle ON clockwise/Coolant ON	■		page 187
M14	Spindle ON counterclockwise/Coolant ON	■		
M30	Same function as M02		■	page 187
M89	Vacant miscellaneous function **or**	■		page 216
	Cycle call, modally effective (depending on machine parameter)		■	
M90	Only in lag mode: Constant contouring speed at corners		■	page 191
M91	Within the positioning block: Coordinates are referenced to machine datum	■		page 188
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		page 188
M94	Reduce display of rotary axis to value under 360°	■		page 204
M97	Machine small contour steps		■	page 193
M98	Machine open contours completely		■	page 194
M99	Blockwise cycle call		■	page 216

M	Effect	Effective at block start	end	Page
M101	Automatic tool change with replacement tool if maximum tool life has expired	■		page 141
M102	Reset M101		■	
M103	Reduce feed rate during plunging to factor F (percentage)	■		page 194
M104	Reactivate the datum as last defined	■		page 190
M105	Machining with second kv factor	■		page 477
M106	Machining with first kv factor	■		
M107	Suppress error message for replacement tools	■		page 141
M108	Reset M107		■	
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)	■		page 196
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)	■		
M111	Reset M109/M110		■	
M114	Automatic compensation of machine geometry when working with tilted axes	■		page 205
M115	Reset M114		■	
M116	Feed rate for angular axes in mm/min	■		page 202
M117	Reset M116		■	
M118	Superimpose handwheel positioning during program run	■		page 198
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	■		page 196
M124	Do not include points when executing non-compensated line blocks	■		page 192
M126	Shortest-path traverse of rotary axes	■		page 203
M127	Reset M126		■	
M128	Maintain the position of the tool tip when positioning with tilted axes (TCPM)	■		page 206
M129	Reset M128		■	
M130	Moving to position in an untilted coordinate system with a tilted working plane	■		page 190
M134	Exact stop at nontangential contour transitions when positioning with rotary axes	■		page 208
M135	Reset M134		■	
M136	Feed rate F in millimeters per spindle revolution	■		page 195
M137	Reset M136		■	
M138	Select tilting axes	■		page 208
M142	Delete modal program information	■		page 201
M143	Delete basic rotation	■		page 201

ISO Function Overview

iTNC 530

M functions	
M00 M01 M02	Stop program/Spindle STOP/Coolant OFF Optional program STOP Stop program run/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP
M08 M09	Coolant ON Coolant OFF
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON
M30	Same function as M02
M89	Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameter)
M90	Only in lag mode: Constant contouring speed at corners
M99	Blockwise cycle call
M91 M92	Within the positioning block: Coordinates are referenced to machine datum Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position
M94	Reduce display of rotary axis to value under 360°
M97 M98	Machine small contour steps Machine open contours completely
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101
M103	Reduce feed rate during plunging to factor F (percentage)
M104	Reactivate the datum as last defined
M105 M106	Machining with second kv factor Machining with first kv factor
M107 M108	Suppress error message for replacement tools Reset M107
M109 M110 M111	Constant contouring speed at tool cutting edge (increase and decrease feed rate) Constant contouring speed at tool cutting edge (feed rate decrease only) Reset M109/M110

M functions	
M114 M115	Automatic compensation of machine geometry when working with tilted axes: Reset M114
M116 M117	Feed rate for angular axes in mm/min Reset M116
M118	Superimpose handwheel positioning during program run
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)
M124	Do not include points when executing non-compensated line blocks
M126 M127	Shortest-path traverse of rotary axes Reset M126
M128 M129	Maintain the position of the tool tip when positioning with tilted axes (TCPM) Reset M128
M130	Moving to position in an untilted coordinate system with a tilted working plane
M134 M135	Exact stop at nontangential contour transitions when positioning with rotary axes Reset M134
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136
M138	Select tilting axes
M142	Delete modal program information
M143	Delete basic rotation
M144 M145	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block Reset M144
M200 M201 M202 M203 M204	Laser cutting: Output programmed voltage directly Laser cutting: Output voltage as a function of distance Laser cutting: Output voltage as a function of speed Laser cutting: Output voltage as a function of time (ramp) Laser cutting: Output voltage as a function of time (pulse)

G functions

Tool Movements

G00 Straight-line interpolation, Cartesian coordinates, rapid traverse
G01 Straight-line interpolation, Cartesian coordinates
G02 Circular interpolation, Cartesian coordinates,
G03 clockwise Circular interpolation, Cartesian coordinates, counterclockwise
G05 Circular interpolation, Cartesian coordinates, without indication of direction
G06 Circular interpolation, Cartesian coordinates, tangential contour connection
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 connection

Chamfer/Rounding/Approach contour/Depart contour

G24* Chamfer with length R
G25* Corner rounding with radius R
G26* Tangential contour approach with tool radius R
G27* Tangential contour departure with tool 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

Cycles for drilling, tapping and thread milling

G83 Pecking
G84 Tapping with a floating tap holder
G85 Rigid tapping
G86 Thread cutting
G200 Drilling
G201 Reaming
G202 Boring
G203 Universal drilling
G204 Back boring
G205 Universal pecking
G206 Tapping with a floating tap holder
G207 Rigid tapping
G208 Bore milling
G209 Tapping with chip breaking

G functions

Cycles for drilling, tapping and thread milling

G262 Thread milling
G263 Thread milling/countersinking
G264 Thread drilling/milling
G265 Helical thread drilling/milling
G267 External thread milling

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

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
G128 Cylindrical surface slot

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
G247 Datum setting

Cycles for multipass milling

G60 Run 3-D data
G230 Multipass milling of plane surfaces
G231 Multipass milling of tilted surfaces

*) Non-modal function

G functions

Touch probe cycles for measuring workpiece misalignment

G400	Basic rotation from two points
G401	Basic rotation from two holes
G402	Basic rotation from two studs
G403	Compensate a basic rotation via a rotary axis
G404	Set basic rotation
G405	Compensating misalignment with the C axis

Touch probe cycles for datum setting

G410	Datum from inside of rectangle
G411	Datum from outside of rectangle
G412	Datum from inside of circle
G413	Datum from outside of circle
G414	Datum in outside corner
G415	Datum in inside corner
G416	Datum circle center
G417	Datum in touch probe axis
G418	Datum in center of 4 holes

Touch Probe Cycles for Automatic Tool Measurement

G55	Measure any coordinate
G420	Measure any angle
G421	Measure hole
G422	Measure cylindrical stud
G423	Measure rectangular pocket
G424	Measure rectangular stud
G425	Measure slot
G426	Measure ridge
G427	Measure any coordinate
G430	Measure circle center
G431	Measure any plane

Touch Probe Cycles for Automatic Tool Measurement

G480	Calibrating the TT
G481	Measure tool length
G482	Measure tool radius
G483	Measure tool length and tool radius

Special Cycles

G04*	Dwell time with F seconds
G36	Oriented spindle stop
G39*	Program call
G62	Tolerance deviation for fast contour milling
G440	Measure axis shift

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

G functions

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)
G79*	Cycle call
G98*	Set label number

*) Non-modal function

Addresses

%	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	Length wear compensation with T
DR	Radius wear compensation with T
E	Tolerance with M112 and M124
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
F	Factor for feed-rate reduction F with M103
G	G functions
H	Polar coordinate angle
H	Rotation angle with G73
H	Tolerance angle with M112
I	Z 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
M	M functions
N	Block number
P	Cycle parameters in machining cycles
P	Value or Q parameter in Q-parameter definition
Q	Q parameter

Addresses	
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

Contour cycles

Sequence of program steps for machining with several tools	
List of subcontour programs	G37 P01 ...
Define contour data	G120 Q1 ...
Define/Call **drill** Contour cycle: pilot drilling Cycle call	G121 Q10 ...
Define/Call **roughing mill** Contour cycle: rough-out Cycle call	G122 Q10 ...
Define/Call **finishing mill** Contour cycle: floor finishing Cycle call	G123 Q11 ...
Define/Call **finishing mill** 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 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+20 Y+30 Z+10	G54 X0 Y0 Z0
Mirror image	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F 0.8	G72 F1
Machining plane	G80 A+10 B+10 C+15	G80

Q-parameter definitions

D	Function
00	Assign
01	Addition
02	Subtraction
03	Multiplication
04	Division
05	Root
06	Sine
07	Cosine
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

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
E-Mail: service@heidenhain.de
Measuring systems ☏ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (711) 952803-0
E-Mail: service.hsf@heidenhain.de

www.heidenhain.de

Ve 00
369 475-21 · 7/2003 · pdf · Subject to change without notice