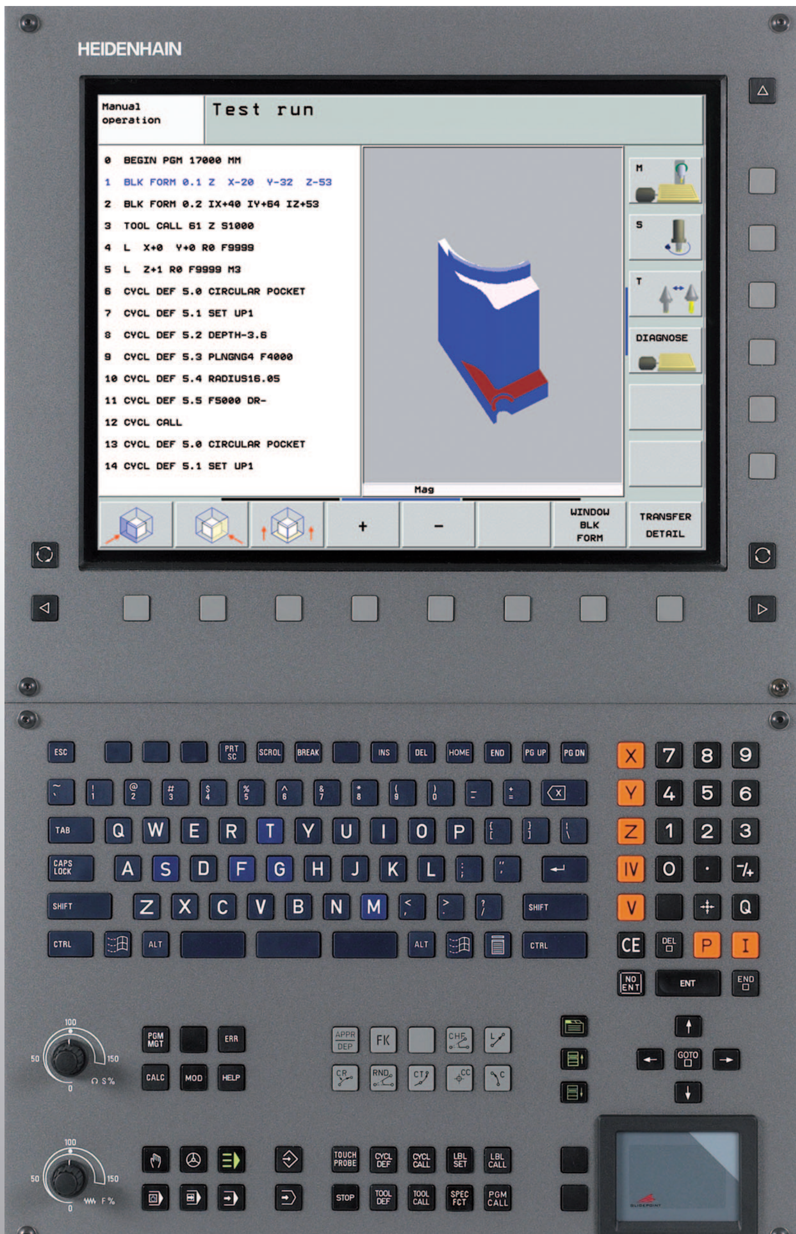
HEIDENHAIN



User's Manual
ISO
Programming

iTNC 530

NC Software
340 490-03
340 491-03
340 492-03
340 493-03
340 494-03

English (en)
10/2006

Controls on the visual display unit

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

Typewriter keyboard for entering letters and symbols

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

Machine operating modes

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

Programming modes

- Programming and Editing
- Test Run

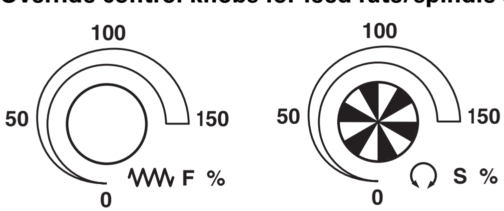
Program/file management, TNC functions

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

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

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

Override control knobs for feed rate/spindle speed



Programming path movements

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

Tool functions

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

Cycles, subprograms and program section repeats

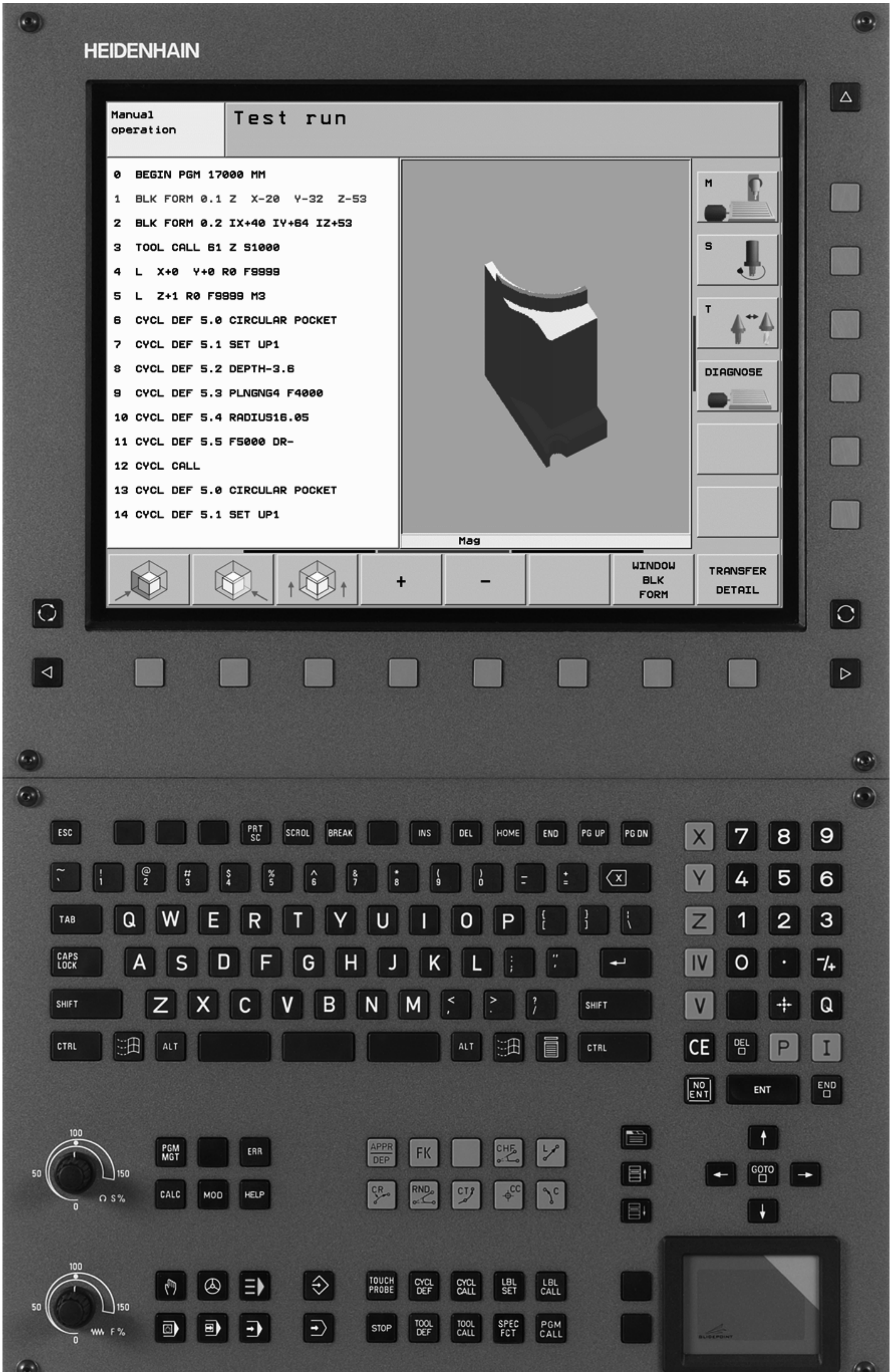
- CYCL DEF, CYCL CALL — Define and call cycles
- LBL SET, LBL CALL — Enter and call labels for subprogramming and program section repeats
- STOP — Program stop in a program
- TOUCH PROBE — Define touch probe cycles

Coordinate axes and numbers: Entering and editing

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

Special functions / smarT.NC

- SPEC FCT — Show special functions
- smarT.NC: Select next tab on form
- smarT.NC: Select first input field in next/ previous frame


HEIDENHAIN
Manual operation
Test run
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 TOOL CALL 61 Z S1000
4 L X+0 Y+0 R0 F9999
5 L Z+1 R0 F9999 M3
6 CYCL DEF 5.0 CIRCULAR POCKET
7 CYCL DEF 5.1 SET UP1
8 CYCL DEF 5.2 DEPTH-3.6
9 CYCL DEF 5.3 PLNGNG4 F4000
10 CYCL DEF 5.4 RADIUS16.05
11 CYCL DEF 5.5 F5000 DR-
12 CYCL CALL
13 CYCL DEF 5.0 CIRCULAR POCKET
14 CYCL DEF 5.1 SET UP1
Mag
M
S
T
DIAGNOSE
+
−
WINDOW BLK FORM
TRANSFER DETAIL

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	340 490-03
iTNC 530 E	340 491-03
iTNC 530	340 492-03
iTNC 530 E	340 493-03
iTNC 530 programming station	340 494-03

The suffix E indicates the export version of the TNC. The export version of the TNC has the following limitations:

- Simultaneous linear movement in up to 4 axes

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

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

- Tool measurement with the TT

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 enhancing your TNC 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 need a copy of this User's Manual. Part number: 533 189-xx

User documentation:

The new smarT.NC operating mode is described in a separate Pilot. Please contact HEIDENHAIN if you require a copy of this Pilot. Part number: 533 191-xx.

Software options

The iTNC 530 features various software options that can be enabled by you or your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Software option 1
Cylinder surface interpolation (Cycles 27, 28, 29 and 39)
Feed rate in mm/min on rotary axes: `M116`
Tilting the machining plane (Cycle 19, `PLANE` function and 3-D ROT soft key in the Manual operating mode)
Circle in 3 axes (with tilted working plane)

Software option 2
Block processing time 0.5 ms instead of 3.6 ms
5-axis interpolation
Spline interpolation
3-D machining: ■ **M114:** Automatic compensation of machine geometry when working with tilted axes ■ **M128:** Maintaining the position of the tool tip when positioning with tilted axes (TCPM) ■ **FUNCTION TCPM:** Maintaining the position of the tool tip when positioning with tilted axes (TCPM) in selectable modes ■ **M144:** Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block ■ Additional parameters **`finishing/roughing`** and **`tolerance for rotary axes`** in Cycle 32 (G62) ■ **LN** blocks (3-D compensation)

DCM software option	Description
Function that monitors areas defined by the machine manufacturer to prevent collisions.	Page 93

DXF Converter software option	Description
Extract contours from DXF files (R12 format).	Page 238

Additional dialog language software option	Description
Function for enabling the conversational languages Slovenian, Slovak, Norwegian, Latvian, Estonian, Korean.	Page 628

Global Program Settings software option	Description
Function for superimposing coordinate transformations in the Program Run modes.	Page 576

AFC software option	Description
Function for adaptive feed-rate control for optimizing the machining conditions during series production.	Page 583

Feature content level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the **Feature Content Level (FCL)** upgrade functions. Functions subject to the FCL are not available simply by updating the software on your TNC.

All upgrade functions are available to you without surcharge when you receive a new machine.

Upgrade functions are identified in the manual with **FCL n**, where **n** indicates the sequential number of the feature content level.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

FCL 3 functions	Description
Touch probe cycle for 3-D probing	Touch Probe Cycles User's Manual
Touch probe cycles for automatic datum setting using the center point of a slot/ ridge	Touch Probe Cycles User's Manual
Feed-rate reduction for the machining of contour pockets with the tool being in full contact with the workpiece	Page 392
PLANE function: Entry of axis angle	Page 480
User documentation as a context-sensitive help system	Page 156
smarT.NC: Programming of smarT.NC and machining can be carried out simultaneously	Conversational Programming User's Manual
smarT.NC: Contour pocket on point pattern	smarT.NC Pilot
smarT.NC: Preview of contour programs in the file manager	smarT.NC Pilot
smarT.NC: Positioning strategy for machining point patterns	smarT.NC Pilot

FCL 2 functions	Description
3-D line graphics	Page 141
Virtual tool axis	Page 92
USB support of block devices (memory sticks, hard disks, CD-ROM drives)	Page 127

FCL 2 functions	Description
Filtering of externally created contours	Conversational Programming User's Manual
Possibility of assigning different depths to each subcontour in the contour formula	Conversational Programming User's Manual
DHCP dynamic IP-address management	Page 603
Touch-probe cycle for global setting of touch-probe parameters	Touch Probe Cycles User's Manual
smarT.NC: Graphic support of block scan	smarT.NC Pilot
smarT.NC: Coordinate transformation	smarT.NC Pilot
smarT.NC: PLANE function	smarT.NC Pilot

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.

Legal information:

This product uses open source software. Further information is available on the control under

- Programming and Editing operating mode
- MOD function
- LEGAL INFORMATION soft key

Functions included in 340 49x-01 new since the predecessor versions 340 422-xx and 340 423-xx

- A new form-based operating mode, smarT.NC, has been introduced. These cycles are described in a separate user's document. In connection with this the TNC operating panel was enhanced. There are some new keys available for quicker navigation within smarT.NC (see "Operating panel" on page 47).
- The single-processor versions supports pointing devices (mice) via the USB 2.0 interface.
- New **CENTERING** cycle (see "CENTERING (Cycle 240)" on page 294)
- New M function M150 for suppressing limit switch messages (see "Suppress limit switch message: M150" on page 269)
- M128 is now also permitted for mid-program startup (see "Mid-program startup (block scan)" on page 568).
- The number of available Q parameters was expanded to 2000 (see "Programming: Q Parameters" on page 505).
- The number of available label numbers was expanded to 1000. Now label names can be assigned as well (see "Labeling Subprograms and Program Section Repeats" on page 490).
- In the Q parameter functions D9 to D12 you can now also assign label names as jump targets (see "If-Then Decisions with Q Parameters" on page 515).
- The current time is also shown in the additional status display window (see "General program information (PGM tab)" on page 54).
- Several columns were added to the tool table (see "Tool table: Standard tool data" on page 183).
- The Test Run can now also be stopped and resumed within machining cycles (see "Running a program test" on page 561).

New functions with 340 49x-02

- DXF files can be opened directly on the TNC, in order to extract contours into a plain-language program (see “Generating Contour Programs from DXF Data (Software Option)” on page 238).
- 3-D line graphics are now available in the Programming and Editing operating mode (see “3-D Line Graphics (FCL 2 Function)” on page 141).
- The active tool-axis direction can now be set as the active machining direction for manual operation (see “Setting the current tool-axis direction as the active machining direction (FCL 2 function)” on page 92).
- The machine manufacturer can now define any areas on the machine for collision monitoring (see “Dynamic Collision Monitoring (Software Option)” on page 93).
- The TNC can now display freely definable tables in the familiar table view or as forms (see “Switching between table and form view” on page 207).
- For contours which you connect via the contour formula, you can now assign separate machining depths for each subcontour (see “SL Cycles with Contour Formulas” on page 415).
- The single-processor version now supports not only pointing devices (mice), but also USB block devices (memory sticks, disk drives, hard disks, CD-ROM drives) (see “USB devices on the TNC (FCL 2 function)” on page 127).

New functions with 340 49x-03

- The AFC function (Adaptive Feed Control) was introduced (see "Adaptive Feed Control Software Option (AFC)" on page 583).
- The global parameter settings function makes it possible to set various transformations and settings in the program run modes (see "Global Program Settings (Software Option)" on page 576).
- The TNC now features a context-sensitive help system, the **TNCguide** (see "The Context-Sensitive Help System TNCguide (FCL3 Function)" on page 156).
- Now you can extract point files from DXF files(see "Selecting and storing machining positions" on page 246).
- Now, in the DXF converter, you can divide or lengthen laterally joined contour elements (see "Dividing, extending and shortening contour elements" on page 245).
- In the **PLANE** function the working plane can now also be defined directly by its axis angle (see "Tilting the working plane through axis angle: PLANE AXIAL (FCL 3 function)" on page 480).
- Now, in Cycle 22 **ROUGH-OUT,** you can define a feed-rate reduction if the tool is cutting on its entire circumference (FCL3 function, see "ROUGH-OUT (Cycle G122)," page 392).
- In Cycle 208 **BORE MILLING,** you can now choose between climb or up-cut milling (see "BORE MILLING (Cycle G208)" on page 309).
- String processing has been introduced in Q parameter programming (see "String Parameters" on page 528).
- A screen saver can be activated through machine parameter 7392 (see "General User Parameters" on page 628).
- The TNC now also supports a network connection over the NFS V3 protocol (see "Ethernet Interface" on page 603).
- The maximum manageable number of tools in a pocket table was increased to 9999 (see "Pocket table for tool changer" on page 190).
- The system time can now be set through the MOD function (see "Setting the System Time" on page 624).

Functions changed in 340 49x-01 since the predecessor versions 340 422-xx/340 423-xx

- The layouts of the status display and additional status display were redesigned (see "Status Displays" on page 51).
- Software 340 490 no longer supports the small resolution in combination with the BC 120 screen (see "Visual display unit" on page 45).
- New key layout of the TE 530 B keyboard unit (see "Operating panel" on page 47)
- The tool types available for selection in the tool table were increased in preparation for future functions.

Functions changed in 340 49x-02

- Access to the preset table was simplified. There are also new possibilities for entering values in the preset table. See table "Manually saving the datums in the preset table".
- In inch-programs, the function M136 (feed rate in 0.1 inch/rev) can no longer be combined with the FU function.
- The feed-rate potentiometers of the HR 420 are no longer switched over automatically when the handwheel is selected. The selection is made via soft key on the handwheel. In addition, the pop-up window for the active handwheel was made smaller, in order to improve the view of the display beneath it (see "Potentiometer settings" on page 72).
- The maximum number of contour elements for SL cycles was increased to 8192, so that much more complex contours can be machined (see "SL Cycles" on page 383).
- **FN16: F-PRINT:** The maximum number of Q-parameter values that can be output per line in the format description file was increased to 32 (Conversational Programming User's Manual).
- The soft keys START and START SINGLE BLOCK in the Program Test mode of operation were switched, so that the soft-key alignment is the same in all modes of operation (Programming and Editing, smarT.NC, Test) (see "Running a program test" on page 561).
- The design of the soft keys was revised completely.

Changed functions with 340 49x-03

- In Cycle 22 you can now define a tool name also for the coarse roughing tool (see “ROUGH-OUT (Cycle G122)” on page 392).
- When running programs in which non-controlled axes are programmed, the TNC now interrupts the program run and displays a menu for returning to the programmed position (see “Programming of noncontrolled axes (counter axes)” on page 565).
- The tool usage file now also includes the total machining time, which serves as the basis for the progress display in percent in the Program Run, Full Sequence mode (see “Tool usage test” on page 571).
- The TNC now also takes the dwell time into account when calculating the machining time in the Test Run mode (see “Measuring the machining time” on page 558).
- Arcs that are not programmed in the active working plane can now also be run as spatial arcs (see “Circular path G02/G03/G05 around circle center I, J” on page 223).
- The EDIT OFF/ON soft key on the pocket table can be deactivated by the machine tool builder (see “Pocket table for tool changer” on page 190).
- The additional status display has been revised. The following improvements have been introduced (see “Additional status displays” on page 53):
  - A new overview page with the most important status displays were introduced.
  - The individual status pages are now displayed as tabs (as in smarT.NC). The individual tabs can be selected over the Page soft keys or with the mouse.
  - The current run time of the program is shown in percent in a moving-bar diagram.
  - The tolerance values set in Cycle 32 are displayed.
  - Active global program settings are displayed, provided that this software option was enabled.
  - The status of the Adaptive Feed Control (AFC) is displayed, provided that this software option was enabled.

Contents

8 Programming: Cycles 281

15 iTNC 530 with Windows 2000 (Option) 653

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 12 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, smarT.NC 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.

The smarT.NC operating mode offers TNC beginners an especially simple possibility to quickly and without much training create structured conversational dialog programs. Separate user documentation is available for smarT.NC.

It is also possible to program the TNCs in ISO format or DNC mode.

You can also enter and test one program while the control is running another (does not apply to smarT.NC).

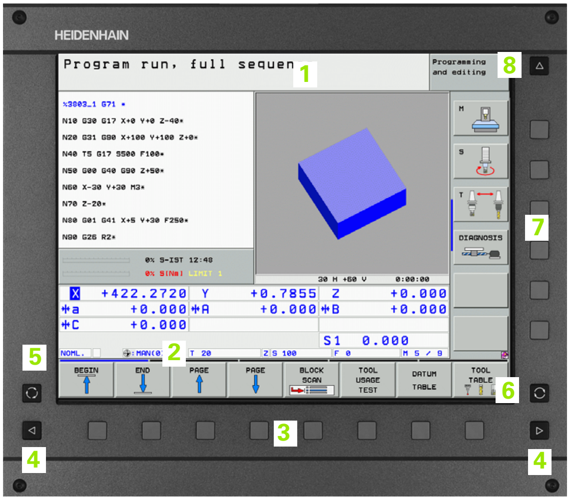
Compatibility

The TNC can run all part programs that were written on HEIDENHAIN controls TNC 150 B and later. In as much as old TNC programs contain OEM cycles, the iTNC 530 must be adapted to them with the PC software CycleDesign. For more information, contact your machine tool builder or HEIDENHAIN.

1.2 Visual Display Unit and Operating Panel

Visual display unit

The TNC is delivered with the BF 150 (TFT) color flat-panel display (see figure at top right).



1 Header

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

2 Soft keys

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

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:

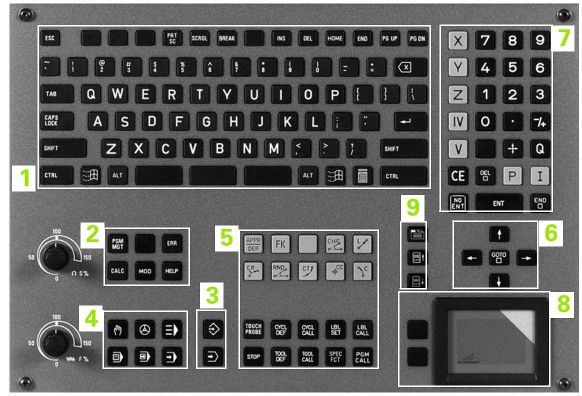
Press the SPLIT SCREEN key: The soft-key row shows the available layout options (see "Modes of Operation," page 48).

PROGRAM + GRAPHICS

Select the desired screen layout.

Operating panel

The TNC is delivered with the TE 530 operating panel. The figure at right shows the controls and displays of the TE 530 keyboard unit.



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 Mousepad: Only for operating the dual-processor version, soft keys and smarT.NC
9 smarT.NC navigation keys

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

Some machine manufacturers do not use the standard operating panel from HEIDENHAIN. Please refer to your machine manual in these cases.

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

1.3 Modes of Operation

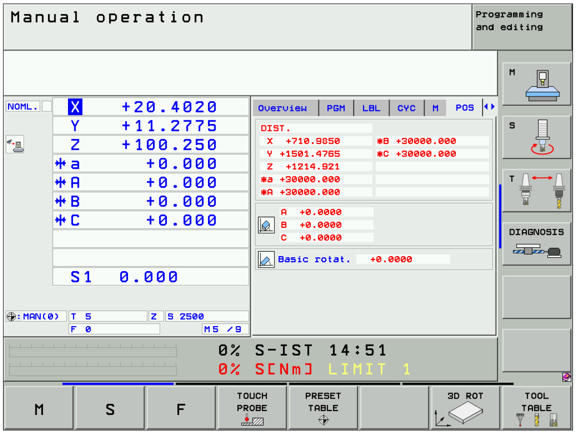
Manual operation and electronic handwheel

The Manual Operation mode is required for setting up the machine tool. In this mode of operation, 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
Positions at left, status display at right	POSITION + STATUS

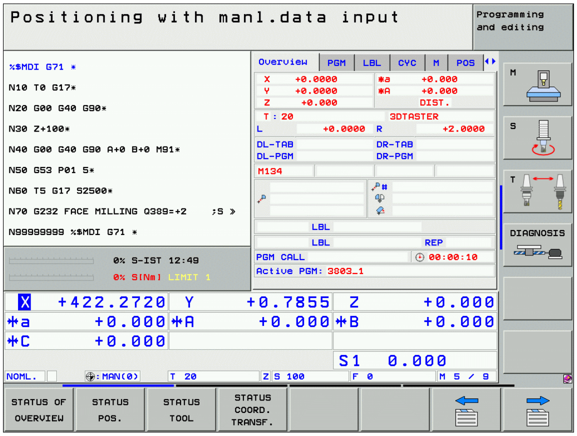


Positioning with Manual Data Input (MDI)

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

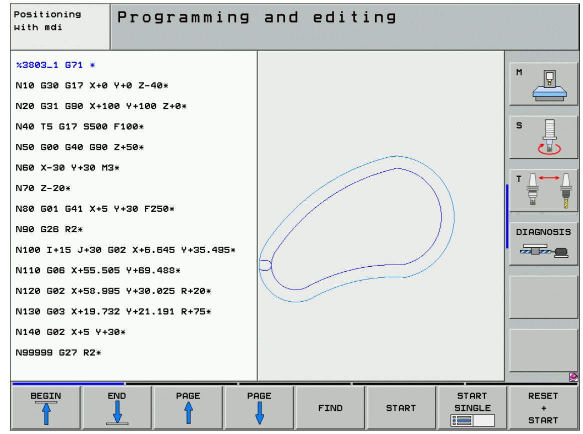
Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Program blocks at left—status display at right	POSITION + 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.

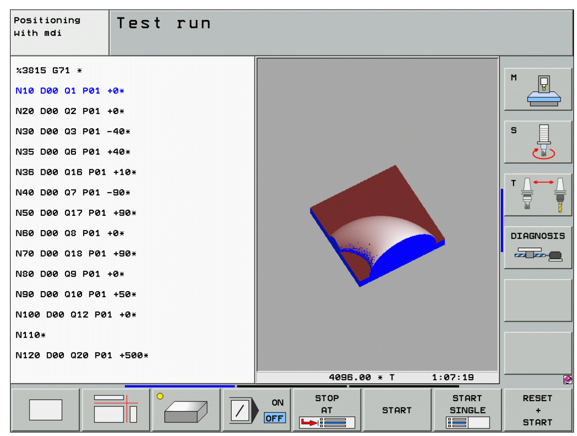


Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program blocks, right: program structure	PROGRAM + SECTS
Left: program, right: programming graphics	PROGRAM + GRAPHICS
Left: program, right: 3-D line graphics	PROGRAM + 3D LINES

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.

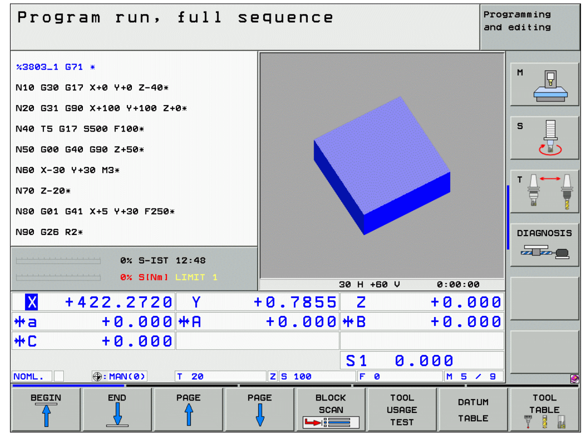


Soft keys for selecting the screen layout: see “Program Run, Full Sequence and Program Run, Single Block,” page 50.

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	PROGRAM + SECTS
Left: program, right: status	PROGRAM + STATUS
Left: program, right: graphics	PROGRAM + 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	PROGRAM + PALLET
Left: pallet table, right: status	PALLET + STATUS
Left: pallet table, right: graphics	PALLET + GRAPHICS

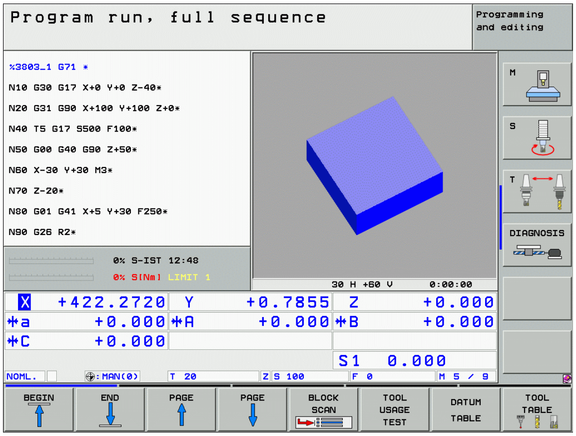
1.4 Status Displays

"General" status display

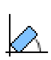
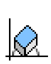
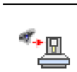
The status display informs you of the current state of the machine tool. It is displayed automatically in the following modes of operation:

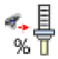
- Program Run, Single Block and Program Run, Full Sequence, except if the screen layout is set to display graphics only, and
- Positioning with Manual Data Input (MDI).

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



Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position.
X Y Z	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information.
F S M	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions.
*	Program run started.
	Axis locked.
	Axis can be moved with the handwheel.
	Axes are moving under a basic rotation.
	Axes are moving in a tilted working plane.
	The **M128** function or **TCPM FUNCTION** is active.
	The **Dynamic Collision Monitoring** function (DCM) is active.

Symbol	Meaning
	The **Adaptive Feed Function** (AFC) is active (software option).
	One or more global program settings are active (software option)
	Number of the active presets from the preset table. If the preset was set manually, the TNC displays the text **MAN** behind the symbol.

Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes except for the Programming and Editing mode of operation.

To switch on the additional status display:

Call the soft-key row for screen layout.

PROGRAM + STATUS

Screen layout with additional status display: In the right half of the screen, the TNC shows the **`Overview`** status form.

To select an additional status display:

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

STATUS POS.

Either select the additional status display, e.g. positions and coordinates, or

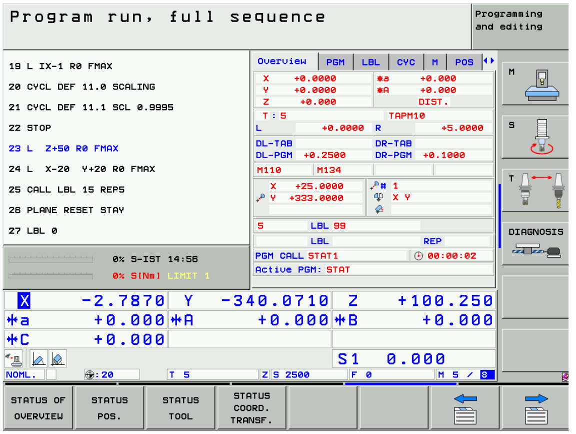
use the soft keys to select the desired view.

With the soft keys or switch-over soft keys, you can choose directly between the available status displays.

Please note that some of the status information described below is not available unless the associated software option is enabled on your TNC.

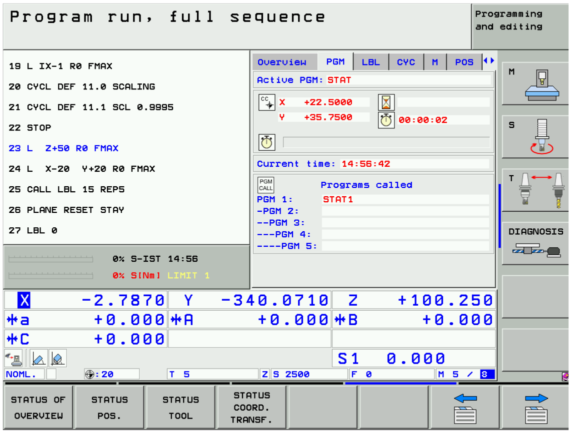
Overview

After switch on, the TNC displays the **Overview** status form, provided that you have selected the PROGRAM+STATUS screen layout (or POSITION + STATUS). The overview form contains a summary of the most important status information, which you can also find on the various detail forms.



Soft key	Meaning
STATUS OF OVERVIEW	Position display in up to 5 axes
	Tool information
	Active M functions
	Active coordinate transformations
	Actives subprogram
	Active program section repeat
	Program called with **PGM CALL**
	Current machining time
	Name of the active main program

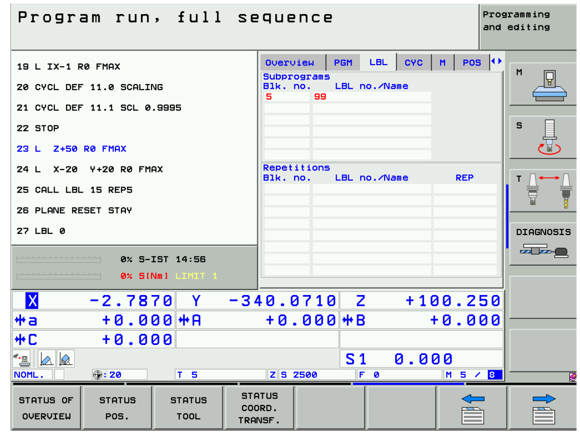
General program information (PGM tab)



Soft key	Meaning
No direct selection possible	Name of the active main program
	Circle center CC (pole)
	Dwell time counter
	Machining time
	Current machining time in percent
	Current time
	Current/programmed contouring feed rate
	Active programs

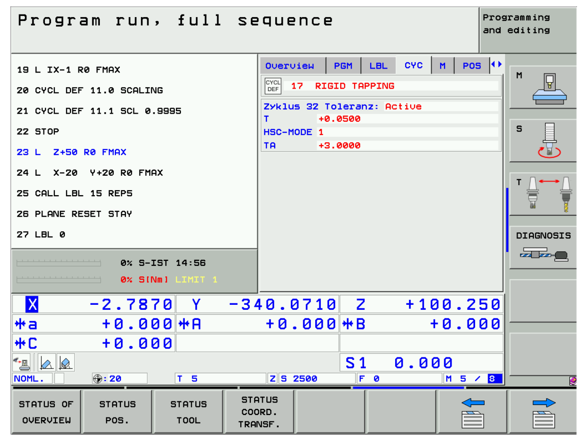
Program section repeat/Subprograms (LBL tab)

Soft key	Meaning
No direct selection possible	Active program section repeats with block number, label number, and number of programmed repeats/repeats yet to be run
	Active subprogram numbers with block number in which the subprogram was called and the label number that was called



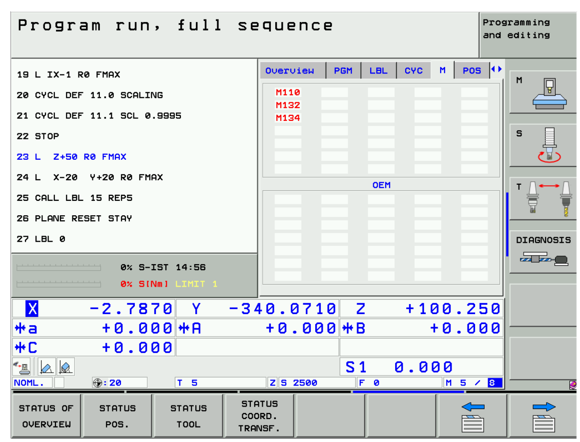
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active machining cycle
	Active values of Cycle G62 Tolerance



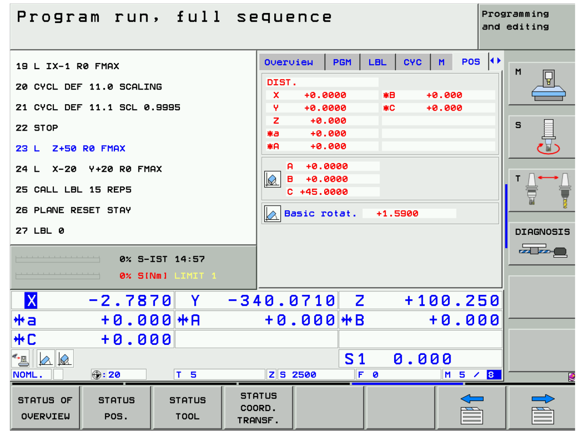
Active miscellaneous functions M (M tab)

Soft key	Meaning
No direct selection possible	List of the active M functions with fixed meaning
	List of the active M functions that are adapted by your machine manufacturer



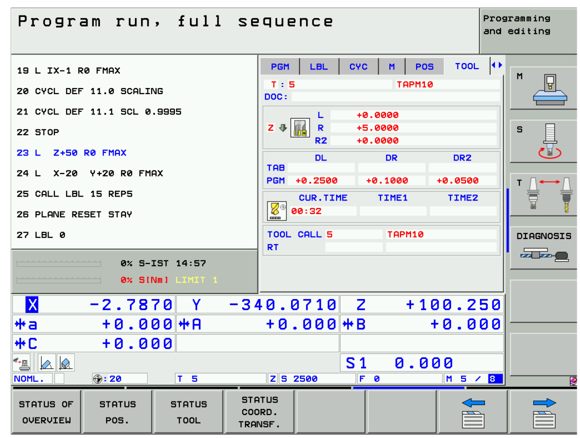
Positions and coordinates (POS tab)

Soft key	Meaning
STATUS POS.	Type of position display, e.g. actual position
	Tilt angle of the working plane
	Angle of a basic rotation



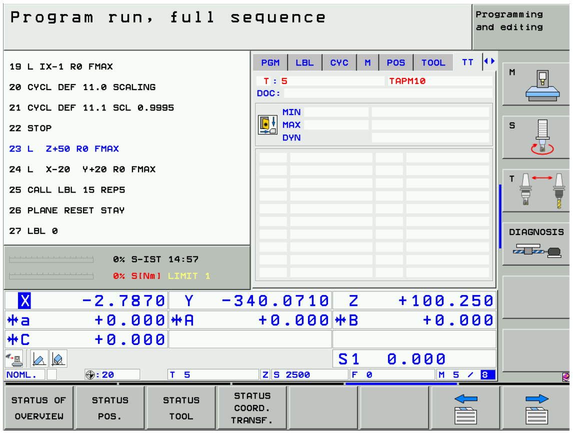
Information on tools (TOOL tab)

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



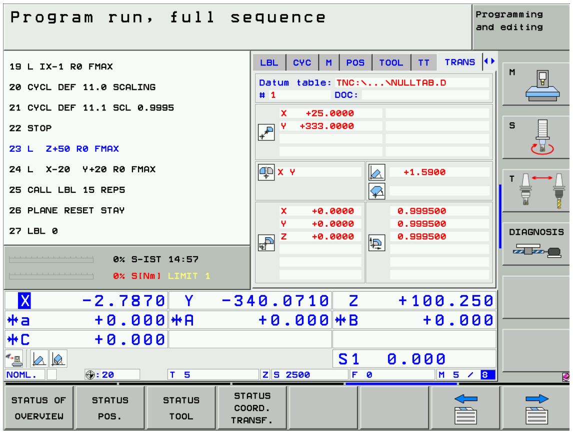
Tool measurement (TT tab)

The TNC only displays the TT tab if the function is active on your machine.



Soft key	Meaning
No direct selection possible	Number of the tool to be measured
	Display whether the tool radius or the tool length is being measured
	MIN and MAX values of the individual cutting edges and the result of measuring the rotating tool (DYN = dynamic measurement)
	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

Coordinate transformations (TRANS tab)



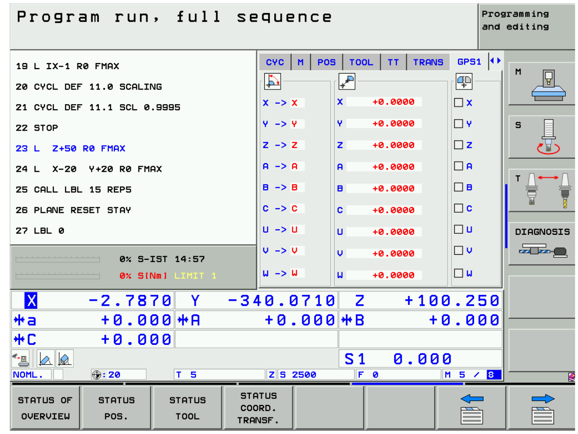
Soft key	Meaning
STATUS COORD. TRANSF.	Name of the active datum table
	Active datum number (#), comment from the active line of the active datum number (**DOC**) from Cycle G53
	Active datum shift (Cycle G54); The TNC displays an active datum shift in up to 8 axes
	Mirrored axes (Cycle G28)
	Active basic rotation
	Active rotation angle (Cycle G73)
	Active scaling factor/factors (Cycles G72); The TNC displays an active scaling factor in up to 6 axes
	Scaling datum

See “Coordinate Transformation Cycles” on page 438.

Global program settings 1 (GPS1 tab, software option)

The TNC only displays the tab if the function is active on your machine.

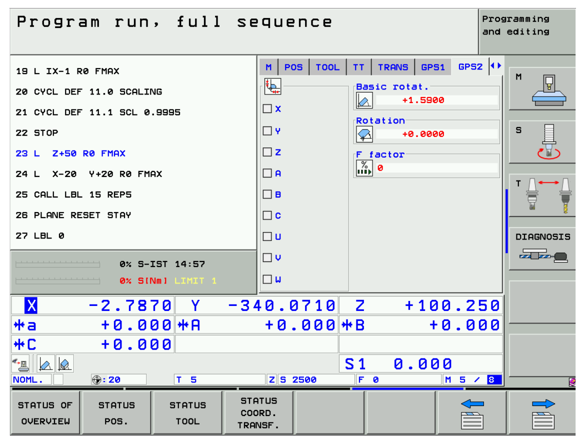
Soft key	Meaning
No direct selection possible	Exchanged axes
	Superimposed datum shift
	Superimposed mirroring



Global program settings 2 (GPS2 tab, software option)

The TNC only displays the tab if the function is active on your machine.

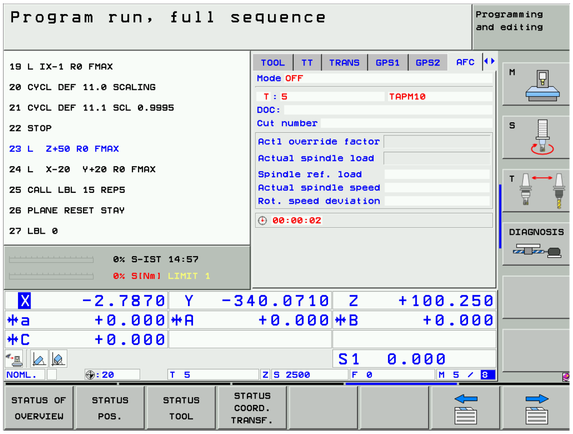
Soft key	Meaning
No direct selection possible	Locked axes
	Superimposed basic rotation
	Superimposed rotation
	Active feed rate factor



Adaptive Feed Control (AFC tab, software option)

> The TNC only displays the **AFC** tab if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Active mode in which the adaptive feed control is run
	Active tool (number and name)
	Cut number
	Current factor of the feed potentiometer in percent
	Active spindle load in percent
	Reference load of the spindle
	Current spindle speed
	Current deviation of the speed
	Current machining time



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.: 533 189-xx.

TS 220 and TS 640 touch trigger probes

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

The TS 640 touch probe features infrared transmission of the triggering signal to the control. This makes it highly convenient for use on machines with automatic tool changers.

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

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) and HR 420 (lower right). You will find a detailed description of HR 420 in Chapter 2 of this manual (see “HR 420 Electronic Handwheel” on page 70).

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.

CONVERT PLC PROGRAM

The PLC program of the TNC is automatically compiled.

RELAY EXT. DC VOLTAGE MISSING

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

MANUAL OPERATION
TRAVERSE REFERENCE POINTS

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

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

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

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

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

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

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 91). The TNC then interpolates the corresponding axes.

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

If available, you can also traverse the axes in the direction of the current tool axis (see "Setting the current tool-axis direction as the active machining direction (FCL 2 function)" on page 92).

If you use this function, then for non-absolute encoders you must confirm the positions of the rotary axes, which the TNC displays in a pop-up window. The position displayed is the last active position of the rotary axes before switch-off.

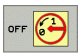
If one of the two functions that were active before is active now, the NC START button has no function. The TNC outputs a corresponding error message.

Switch-off

iTNC 530 with Windows 2000: See “Switching Off the iTNC 530,” page 658.

To avoid losing data at switch-off, you need to shut down the operating system as follows:

- Select the Manual Operation mode.



- Select the function for shutting down, confirm again with the YES soft key.
- When the TNC displays the message **`Now 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

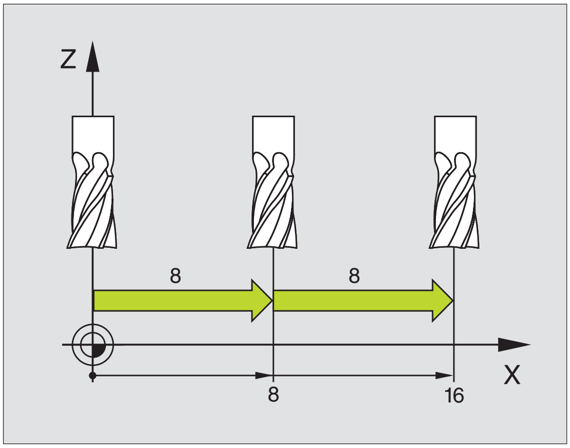
X I — Move the axis continuously: Press and hold the machine axis direction button, then press the machine START button.

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 Fsoft key (see "Spindle Speed S, Feed Rate F and Miscellaneous Functions M," page 76).

Incremental jog positioning

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



Select the Manual Operation or Electronic Handwheel mode.

Shift the soft-key row.

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.

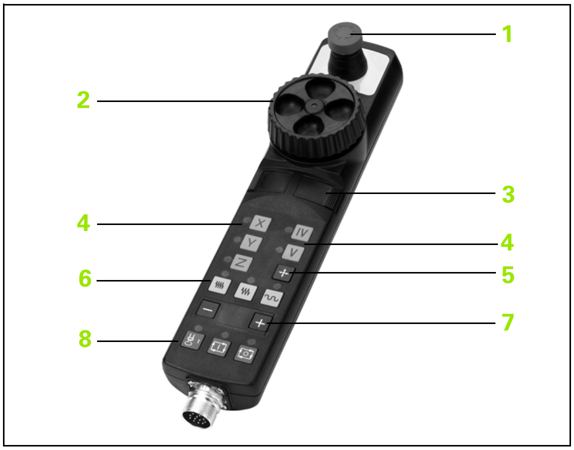
The maximum permissible value for infeed is 10 mm.

Traversing with the HR 410 electronic handwheel

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

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

The HR 410 handwheel features the following operating elements:



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

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

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

Procedure:

Select the Electronic Handwheel operating mode.

Press and hold a permissive button.

Select the axis.

Select the feed rate.

Move the active axis in the positive direction, or

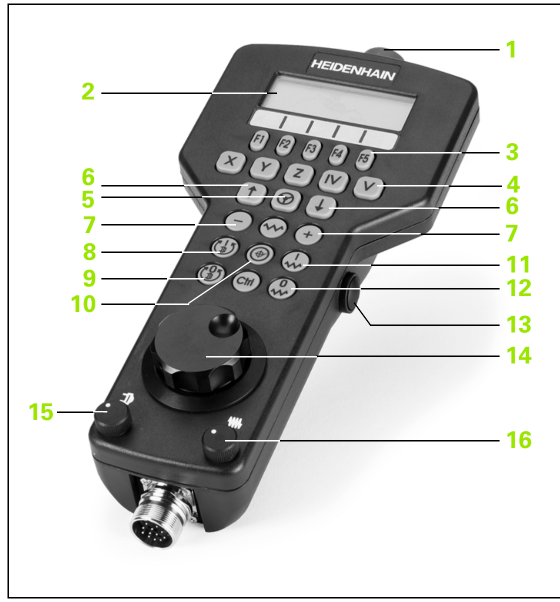
Move the active axis in the negative direction.

HR 420 Electronic Handwheel

Unlike the HR 410, the HR 420 portable handwheel is equipped with a display. In addition, you can run important setup functions through the handwheel soft keys, e.g. setting datums or entering and running M functions.

As soon as you press the handwheel activation key, it activates the handwheel and deactivates the control panel. This is indicated by a pop-up window on the TNC screen.

The HR 420 handwheel features the following operating elements:



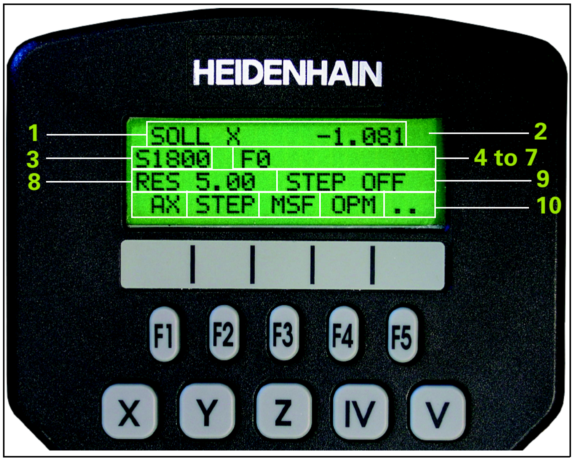
1 EMERGENCY OFF button
2 Handwheel display for status display and function selection
3 Soft keys
4 Axis address keys
5 Handwheel activation key
6 Arrow keys for definition of handwheel sensitivity
7 Direction key by which the TNC moves the selected axis
8 Switch on the spindle (machine-specific M function)
9 Switch off the spindle (machine-specific M function)
10 NC-block creation key
11 NC start
12 NC stop
13 Permissive button
14 Handwheel
15 Spindle speed potentiometer
16 Feed rate potentiometer

If **M118** is active, it is even possible to move the machine axes with the handwheel during the program run.

Your machine manufacturer can make additional functions of the HR 420 available. Refer to your machine manual.

Display

The handwheel display has four lines (see figure). The TNC shows there the following information:



1 **NOML X+1.563:** Type of position display and position of the selected axis
2 *****: STIB (control is in operation)
3 **S1000:** Current spindle speed
4 **F500:** Feed rate at which the selected axis is moving
5 **E:** There is an error
6 **3D:** Tilted-working-plane function is active
7 **2D:** Basic rotation function is active
8 **RES 5.0:** Active handwheel resolution. Distance in mm/rev (°/rev for rotary axes) that the selected axis moves for one handwheel revolution
9 **STEP ON** or **OFF:** Incremental jog active or inactive. If the function is active, the TNC also displays the active jog increment
10 Soft key row: Selection of various functions, described in the following sections

Select the axis to be moved

You can activate directly through the axis address keys the principal axes X, Y, Z and two other axes defined by the machine tool builder. If your machine has more axes, proceed as follows.

- Press the handwheel soft key F1 **(AX)**: The TNC displays all active axes on the handwheel display. The active axis blinks.
- Select the desired axis with the handwheel soft key F1 **(->)** or F2 **(<-)** and confirm your selection with F3 **(OK).**

Set the handwheel sensitivity

The handwheel sensitivity defines the distance that an axis is to move per handwheel revolution. The sensitivity levels are ready-defined and are selectable with the handwheel arrow keys (unless incremental jog is not active).

Selectable sensitivity levels: 0.01/0.02/0.05/0.1/0.2/0.5/1/2/5/10/20 [mm/revolution or degrees/revolution]

Moving the axes

Activate the handwheel: Press the handwheel key on the HR 420. Now the TNC is operable only through the HR 420. A pop-up window stating such appears on the TNC screen.

Select the desired operating mode via the OPM soft key, if necessary (see “Changing the modes of operation” on page 74).

ENT

If required, press and hold the permissive button.

X

Use the handwheel to select the axis to be moved. Select the additional axes via soft key.

+

Move the active axis in the positive direction, or

–

Move the active axis in the negative direction.

Deactivate the handwheel: Press the handwheel key on the HR 420. Now the TNC can be operated through the control panel.

Potentiometer settings

The potentiometers of the machine operating panel continue to be active after you have activated the handwheel. If you want to use the potentiometers on the handwheel, proceed as follows:

- Press the CTRL and Handwheel keys in the HR 420. The TNC shows the soft-key menu for selecting the potentiometers on the handwheel display.
- Press the HW soft key to activate the handwheel potentiometers.

If you have activated the potentiometers on the handwheel, you must reactivate the potentiometers of the machine operating panel before deselecting the handwheel. Proceed as follows:

- Press the CTRL and Handwheel keys in the HR 420. The TNC shows the soft-key menu for selecting the potentiometers on the handwheel display.
- Press the KBD soft key to activate the potentiometers of the machine operating panel.

Incremental jog positioning

With incremental jog positioning the TNC moves the currently active handwheel axis by a preset distance defined by you.

- Press the handwheel soft key F2 **(STEP)**.
- Activate incremental jog positioning: Press handwheel soft key 3 (**ON**).
- Select the desired jog increment by pressing the F1 or F2 key. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. If in addition you press the Ctrl key, the counting increment increases to 1. The smallest possible jog increment is 0.0001 mm. The largest possible is 10 mm.
- Confirm the selected jog increment with soft key 4 **(OK).**
- With the + or – handwheel key, move the active handwheel axis in the corresponding direction.

Entering miscellaneous functions M

- Press the handwheel soft key F3 **(MSF).**
- Press the handwheel soft key F1 **(M).**
- Select the desired M function number by pressing the F1 or F2 key.
- Execute the M function with the NC start key.

Entering the spindle speed S

- Press the handwheel soft key F3 **(MSF).**
- Press the handwheel soft key F2 **(S).**
- Select the desired speed by pressing the F1 or F2 key. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. If in addition you press the Ctrl key, the counting increment increases to 1000.
- Activate the new speed S with the NC start key.

Entering the feed rate F

- Press the handwheel soft key F3 **(MSF).**
- Press the handwheel soft key F3 **(F).**
- Select the desired feed rate by pressing the F1 or F2 key. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. If in addition you press the Ctrl key, the counting increment increases to 1000.
- Confirm the new feed rate F with the handwheel soft key F3 **(OK).**

Workpiece Presetting

- Press the handwheel soft key F3 **(MSF).**
- Press the handwheel soft key F4 **(PRS).**
- If required, select the axis in which the datum is to be set.
- Reset the axis with the handwheel soft key F3 **(OK),** or with F1 and F2 set the desired value and then confirm with F3 **(OK).** By also pressing the Ctrl key, you can increase the counting increment to 10.

Changing the modes of operation

With the handwheel soft key F4 **(OPM),** you can use the handwheel to switch the mode of operation, provided that the current status of the control allows a mode change.

- Press the handwheel soft key F4 **(OPM).**
- Select the desired operating mode by handwheel soft key.
  - MAN: Manual Operation
  - MDI: Positioning with Manual Data Input
  - SGL: Program Run, Single Block
  - RUN: Program Run, Full Sequence

Generating a complete L Block

> Use the MOD function to define the axis values to be taken into an NC block (see “Selecting the Axes for Generating L Blocks” on page 619).
>
> If no axes are selected, the TNC displays the error message **No axes selected.**

- Select the **Positioning with MDI** operating mode.
- If required, use the arrow keys on the TNC keyboard to select the NC block after which the new L block is to be inserted.
- Actuate handwheel.
- Press the Generate-NC-block handwheel key: The TNC inserts a complete L block containing all axis positions selected through the MOD function.

Features in the Program Run modes of operation

You can use the following functions in the Program Run modes of operation:

- NC start (handwheel NC-start key)
- NC stop (handwheel NC-stop key)
- After the NC-stop key has been pressed: Internal stop (handwheel soft keys **MOP** and then **STOP**)
- After the NC-stop key has been pressed: Manual axis traverse (handwheel soft keys **MOP** and then **MAN**)
- Returning to the contour, after the axes were moved manually during a program interruption (handwheel soft keys **MOP** and then **REPO**). Operation is by handwheel soft keys, which function similarly to the control-screen soft keys (see "Returning to the contour" on page 570)
- On/off switch for the Tilted Working Plane function (handwheel soft keys **MOP** and then **3D**)

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.

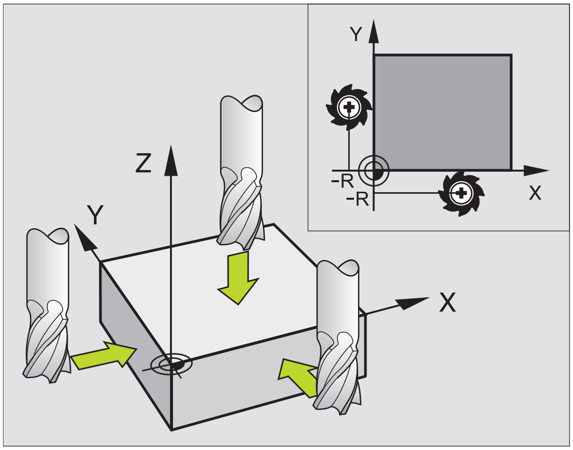
Preparation

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

Datum setting with axis keys

Fragile workpiece?

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



Select the **Manual Operation** mode.

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

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

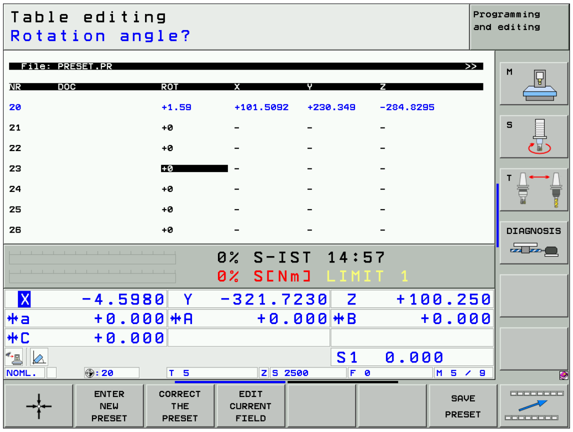
DATUM SET Z=

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

Repeat the process for the remaining axes.

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

Datum management with the preset table



You should definitely use the preset table if:

- Your machine is equipped with rotary axes (tilting table or swivel head) and you work with the function for tilting the working plane
- Your machine is equipped with a spindle-head changing system
- Up to now you have been working with older TNC controls with REF-based datum tables
- You wish to machine identical workpieces that are differently aligned

The preset table can contain any number of lines (datums). To optimize the file size and the processing speed, you should use only as many lines as you need for datum management.

For safety reasons, new lines can be inserted only at the end of the preset table.

Saving the datums in the preset table

The preset table has the name **PRESET.PR**, and is saved in the directory **TNC:**. **PRESET.PR** is editable only in the **Manual Operation** and **Electronic Handwheel** modes. In the Programming and Editing mode you can only read the table, not edit it.

It is permitted to copy the preset table into another directory (for data backup). Lines that were written by your machine tool builder are also always write-protected in the copied tables. You therefore cannot edit them.

Never change the number of lines in the copied tables! That could cause problems when you want to reactivate the table.

To activate the preset table copied to another directory you have to copy it back to the directory **TNC:**.

There are several methods for saving datums and/or basic rotations in the preset table:

- Through probing cycles in the **Manual Operation** or **Electronic Handwheel** modes (see User's Manual, Touch Probe Cycles, Chapter 2)
- Through the probing cycles 400 to 402 and 410 to 419 in automatic mode (see User's Manual, Touch Probe Cycles, Chapter 3)
- Manual entry (see description below)

Basic rotations from the preset table rotate the coordinate system about the preset, which is shown in the same line as the basic rotation.

When setting a preset, the TNC checks whether the position of the tilting axes match the corresponding values of the 3D ROT menu (depending on the MP setting). Therefore:

- If the "Tilt working plane" function is not active, the position displays for the rotary axes must = 0° (zero the rotary axes if necessary).
- If the "Tilt working plane" function is active, the position displays for the rotary axes must match the angles entered in the 3D ROT menu.

The machine manufacturer can lock any lines in the preset table in order to place fixed datums there (e.g. a center point for a rotary table). Such lines in the preset table are shown in a different color (default: red).

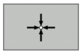
The line 0 in the preset table is write protected. In line 0, the TNC always saves the datum that you most recently set manually via the axis keys or via soft key. If the datum set manually is active, the TNC displays the text **PR MAN(0)** in the status display.

If you automatically set the TNC display with the touch-probe cycles for presetting, then the TNC does not store these values in line 0.

Manually saving the datums in the preset table

In order to set datums in the preset table, proceed as follows:

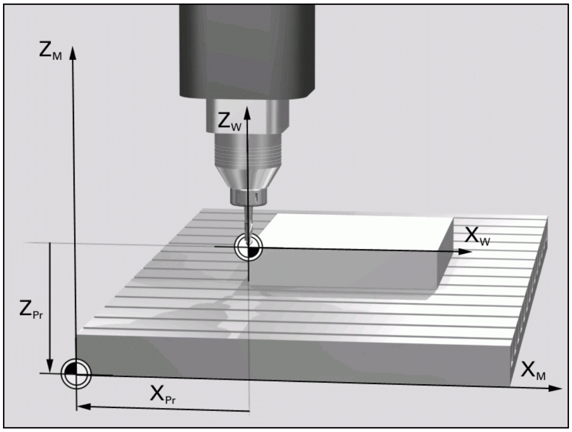
	Select the **Manual Operation** mode.
X Y Z	Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly.
PRESET TABLE	Displaying the preset table: The TNC opens the preset table and sets the cursor to the active table row.
CHANGE PRESET	Select functions for entering the presets: The TNC displays the available possibilities for entry in the soft-key row. See the table below for a description of the entry possibilities.
	Select the line in the preset table which you want to change (the line number is the preset number).
	If needed, select the column (axis) in the preset table which you want to change.
CORRECT THE PRESET	Use the soft keys to select one of the available entry possibilities (see the following table).

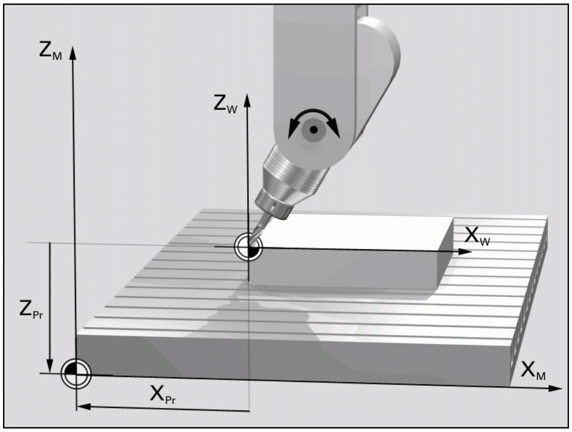
Function	Soft key
Directly transfer the actual position of the tool (the measuring dial) as the new datum: This function only saves the datum in the axis which is currently highlighted.	
Assign any value to the actual position of the tool (the measuring dial): This function only saves the datum in the axis which is currently highlighted. Enter the desired value in the pop-up window.	ENTER NEW PRESET
Incrementally shift a datum already stored in the table: This function only saves the datum in the axis which is currently highlighted. Enter the desired corrective value with the correct sign in the pop-up window.	CORRECT THE PRESET
Directly enter the new datum without calculation of the kinematics (axis-specific). Only use this function if your machine has a rotary table, and you want to set the datum to the center of the rotary table by entering 0. This function only saves the datum in the axis which is currently highlighted. Enter the desired value in the pop-up window.	ENTER VALUE DIRECTLY
Write the currently active datum to a selectable line in the table: This function saves the datum in all axes, and then activates the appropriate row in the table automatically.	SAVE PRESET

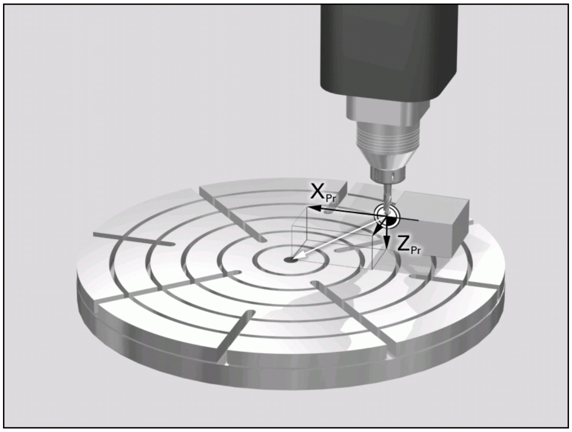
Explanation of values saved in the preset table

- Simple machine with three axes without tilting device
  The TNC saves in the preset table the distance from the workpiece datum to the reference point (with the correct algebraic sign).
- Machine with swivel head
  The TNC saves in the preset table the distance from the workpiece datum to the reference point (with the correct algebraic sign).
- Machine with rotary table
  The TNC saves in the preset table the distance from the workpiece datum to the center of the rotary table (with the correct algebraic sign).
- Machine with rotary table and swivel head
  The TNC saves in the preset table the distance from the workpiece datum to the center of the rotary table.

Keep in mind that moving an indexing feature on your machine table (realized by changing the kinematic description) requires you to redefine any workpiece-based presets.







Editing the preset table

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
Select the functions for preset entry	CHANGE PRESET
Activate the datum of the selected line of the preset table	ACTIVATE PRESET
Add the entered number of lines to the end of the table (2nd soft-key row)	APPEND N LINES
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD
Reset the selected line: The TNC enters – in all columns (2nd soft-key row)	RESET LINE
Insert a single line at the end of the table (2nd soft-key row)	INSERT LINE
Delete a single line at the end of the table (2nd soft-key row)	DELETE LINE

Activating the datum from the preset table in the Manual Operation mode

When activating a datum from the preset table, the TNC resets all coordinate transformations that were activated with the following cycles:

- Cycle 7, Datum Shift
- Cycle 8, Mirroring
- Cycle 10, Rotation
- Cycle 11, Scaling
- Cycle 26, Axis-Specific Scaling

However, the coordinate transformation from Cycle 19, Tilted Working Plane, remains active.

Select the **Manual Operation** mode.

PRESET TABLE — Display the preset table.

Select the datum number that you want to activate, or

GOTO 4 ENT — With the GOTO key, select the datum number that you want to activate. Confirm with the ENT key.

ACTIVATE PRESET — Activate the preset.

EXECUTE — Confirm activation of the datum. The TNC sets the display and—if defined—the basic rotation.

END — Leave the preset table.

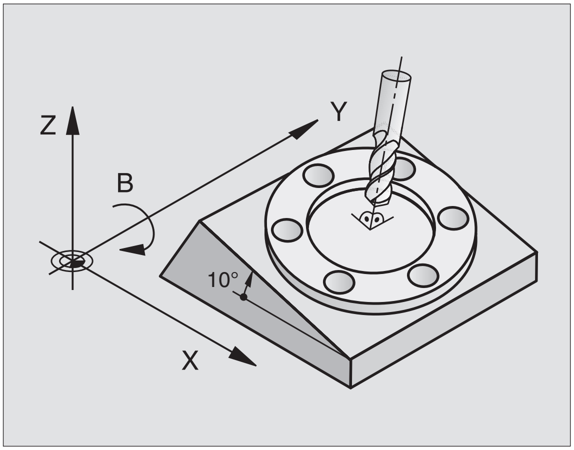
Activating the datum from the preset table in an NC program

To activate datums from the preset table during program run, use Cycle 247. In Cycle 247 you define the number of the datum that you want to activate (see “DATUM SETTING (Cycle G247)” on page 443).

2.5 Tilting the Working Plane (Software Option 1)

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 three functions available for tilting the working plane:

- 3-D ROT soft key in the Manual Operation mode and Electronic Handwheel mode, see "Activating manual tilting," page 91.
- Tilting under program control, Cycle 19 **WORKING PLANE,** in the part program (see "WORKING PLANE (Cycle G80, software option1)" on page 448).
- Tilting under program control, **PLANE** function in the part program (see "The PLANE Function: Tilting the Working Plane (Software Option 1)" on page 464).

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 an L block.
  - The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system **does not rotate.** If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in Z+ direction.
  - In calculating the transformed coordinate system, the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called “translational” components).
- **Machine with swivel head**
  - You must bring the tool into the desired position for machining by positioning the swivel head, for example with an L block.
  - The position of the transformed tool axis changes in relation to the machine-based coordinate system. Thus if you rotate the swivel head 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 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 behavior of the TNC during datum setting depends on the settings of Machine Parameter 7500 in your kinematics table:

- **MP 7500, bit 5=0**
  With an active tilted working plane, the TNC checks during datum setting in the X, Y and Z axes whether the current coordinates of the rotary axes agree with the tilt angles that you defined (3D-ROT menu). If the tilted working plane function is not active, the TNC checks whether the rotary axes are at 0° (actual positions). If the positions do not agree, the TNC will display an error message.
- **MP 7500, bit 5=1**
  The TNC does not check whether the current coordinates of the rotary axes (actual positions) agree with the tilt angles that you defined.

Always set a reference point in all three reference axes.

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

If you use a rotary table to align the workpiece, for example with probing cycle 403, you must set the table position value to zero after alignment and before setting the datum in the linear axes X, Y and Z. The TNC will otherwise display an error message. Cycle 403 provides you with an input parameter for this purpose (see User's Manual for Touch Probe Cycles, "Basic Rotation Compensation via Rotary Axis").

Datum setting on machines with spindle-head changing systems

If your machine is equipped with a spindle head changer, you should use the preset table to manage your datums. Datums saved in preset tables account for the active machine kinematics (head geometry). If you exchange heads, the TNC accounts for the new head dimensions so that the active datum is retained.

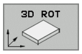
Position display in a tilted system

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 probing function for basic rotation is not available if you have activated the working plane function in the Manual Operation mode.
- PLC positioning (determined by the machine tool builder) is not possible.

Activating manual tilting



To select manual tilting, press the 3-D ROT soft key.

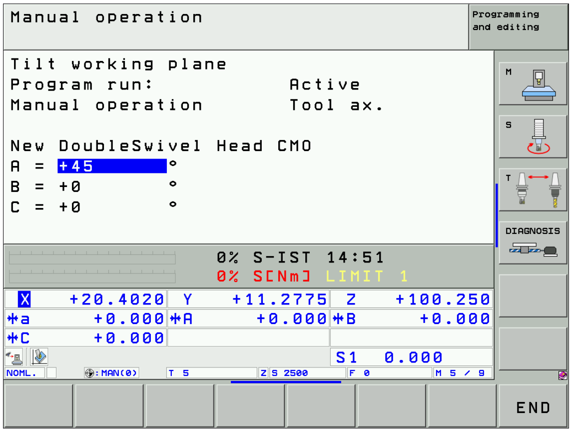
Use the arrow keys to move the highlight to the **Manual Operation** menu item.

To activate manual tilting, press the ACTIVE soft key.

Use the arrow keys to position the highlight on the desired rotary axis.

Enter the tilt angle.

To conclude entry, press the END key.



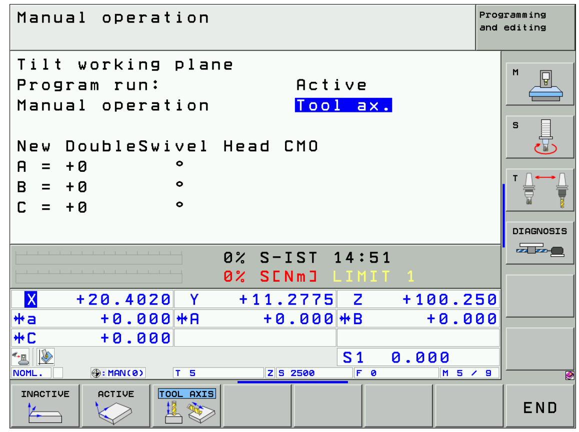
To reset the tilting function, set the desired operating modes in the menu “Tilt working plane” to inactive.

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 use Cycle 19 **WORKING PLANE** or the **PLANE** function in the machining program, the angle values defined there are in effect. Angle values entered in the menu will be overwritten.

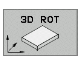
Setting the current tool-axis direction as the active machining direction (FCL 2 function)

This function must be enabled by your machine manufacturer. Refer to your machine manual.



In the Manual Operation and Electronic Handwheel modes of operation you can use this function to move the tool via the external direction keys or with the handwheel in the direction that the tool axis is currently pointed. Use this function if

- You want to retract the tool in the direction of the tool axis during program interrupt of a 5-axis machining program.
- You want to machine with an inclined tool using the handwheel or the external direction keys in the Manual Operation mode.



To select manual tilting, press the 3-D ROT soft key.

Use the arrow keys to move the highlight to the `Manual Operation` menu item.

To activate the current tool-axis direction as the active machining direction, press the TOOL AXIS soft key.

To conclude entry, press the END key.

To reset the tilting function, set the `Manual Operation` menu item in the “Tilt working plane” menu to inactive.

The symbol appears in the status display when the **Move in tool-axis direction** function is active.

The main axis of the active working plane (X with tool axis Z) is always in the machine’s permanent main plane (Z/X with tool axis Z).

This function is even available when you interrupt program run and want to move the axes manually.

2.6 Dynamic Collision Monitoring (Software Option)

Function

The Dynamic Collision Monitoring **(DCM)** must be adapted by the machine manufacturer for the TNC and for the machine. Refer to your machine manual.

The machine manufacturer can define any objects that are monitored by the TNC during all machining operations. If two objects monitored for collision approach each other within a defined distance, the TNC outputs an error message.

The TNC also monitors the current tool with the length and radius entered in the tool table for collision (assuming a cylindrical tool).

Please note that for certain tools (such as face milling cutters), the diameter that would cause a collision can be greater than the dimensions defined in the tool-compensation data.

The dynamic collision monitoring is active in all machine operating modes, and is indicated by a symbol in the operating mode display.

Collision monitoring in the manual operating modes

In the **`Manual Operation`** and **`Electronic Handwheel`** operating modes, the TNC stops a motion if two objects monitored for collision approach each other within a specified distance. In addition, the TNC reduces the feed rate significantly when the distance to the limit value triggering the error is less than 5 mm.

There are three zones determining the TNC’s corrective behavior:

- Early warning: Two objects monitored for collision are within **14 mm** of each other
- Warning: Two objects monitored for collision are within **8 mm** of each other
- Error: Two objects monitored for collision are within **2 mm** of each other

Early warning zone

Two objects monitored for collision are within **12 to 14 mm** of each other The error message displayed (the machine manufacturer determines the exact text) always starts with this text string: **`]--[`**

- Acknowledge the error message with the CE key.
- Manually traverse the axes out of the danger zone. Pay attention to the direction of traverse.
- If applicable, remove the cause of the collision message.

Warning zone

Two objects monitored for collision are within **6 to 8 mm** of each other The error message displayed (the machine manufacturer determines the exact text) always starts with this text string: **`]-[`**

- Acknowledge the error message with the CE key.
- Manually traverse the axes out of the danger zone. Pay attention to the direction of traverse.
- If applicable, remove the cause of the collision message.

Error zone

Two objects monitored for collision are less than **2 mm** from each other The error message displayed (the machine manufacturer determines the exact text) always starts with this text string: **`][`**. In this state you can only traverse the axes after deactivating collision monitoring:

- To select the menu for deactivating collision monitoring, press the Collision Monitoring soft key (rear soft-key row).
- Use the arrow keys to select the **`Manual Operation`** menu item.
- To deactivate collision monitoring, press the ENT key, and the symbol for collision monitoring in the operating mode display starts to blink.
- Acknowledge the error message with the CE key.
- Manually traverse the axes out of the danger zone. Pay attention to the direction of traverse.
- If applicable, remove the cause of the collision message.
- To reactivate collision monitoring, press the ENT key, and the symbol for collision monitoring in the operating mode display remains on again.

Collision monitoring in Automatic operation

The handwheel superimposition function with M118 is not possible in combination with collision monitoring.

If collision monitoring is on, the TNC shows the symbol in the position display.

If you have deactivated the collision monitoring, the symbol for collision monitoring flashes in the operating-mode bar.

The M140 (see “Retraction from the contour in the tool-axis direction: M140” on page 265) and M150 (see “Suppress limit switch message: M150” on page 269) functions might cause non-programmed movements if the TNC detects a collision when executing these functions!

The TNC monitors motions blockwise, i.e. it outputs a warning in the block which would cause a collision, and interrupts program run. A reduction of the feed rate, as with Manual Operation, does not occur.

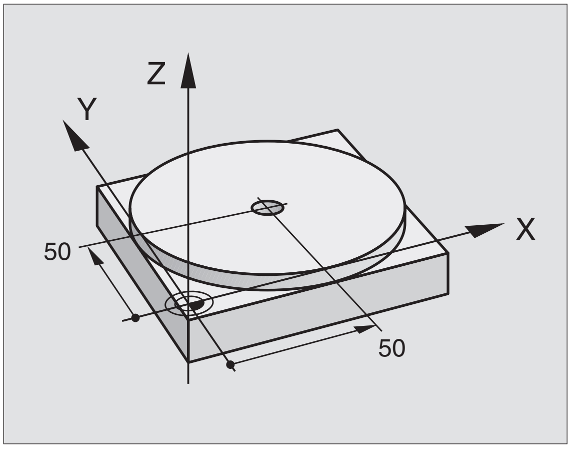
3

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 key.

Limitation

The programming graphics and program run graphics cannot be used. The $MDI file must not contain a program call (%).

Example 1

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

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

`%$MDI G71 *`	
`N10 G99 T1 L+0 R+5 *`	Define tool: zero tool, radius 5
`N20 T1 G17 S2000 *`	Call tool: tool axis Z
	Spindle speed 2000 rpm
`N30 G00 G40 G90 Z+200 *`	Retract tool (rapid traverse)
`N40 X+50 Y+50 M3 *`	Move the tool at rapid traverse to a position above the hole
	Spindle on
`N50 G01 Z+2 F2000 *`	Position tool to 2 mm above hole
`N60 G200 DRILLING *`	Define Cycle G200 Drilling
`Q200=2 ;SET-UP CLEARANCE`	Set-up clearance of the tool above the hole
`Q201=-20 ;DEPTH`	Total hole depth (algebraic sign=working direction)
`Q206=250 ;FEED RATE FOR PLNGNG`	Feed rate for pecking
`Q202=10 ;PLUNGING DEPTH`	Depth of each infeed before retraction
`Q210=0 ;DWELL TIME AT TOP`	Dwell time at top for chip release (in seconds)
`Q203=+0 ;SURFACE COORDINATE`	Workpiece surface coordinate
`Q204=50 ;2ND SET-UP CLEARANCE`	Position after the cycle, with respect to Q203
`Q211=0.5 ;DWELL TIME AT DEPTH`	Dwell time in seconds at the hole bottom
`N70 G79 *`	Call Cycle G200 PECKING
`N80 G00 G40 Z+200 M2 *`	Retract the tool
`N9999999 %$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 219), Cycle **G200** DRILLING (see “DRILLING (Cycle G200)” on page 296).

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 rotary table axis, enter the rotation angle you wrote down and set the feed rate. For example:
G01 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.
EXECUTE	Copy the file.
END	To close the file manager, press the END soft key.

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

If you wish to delete $MDI, then

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

For further information, see "Copying a single file," page 117.

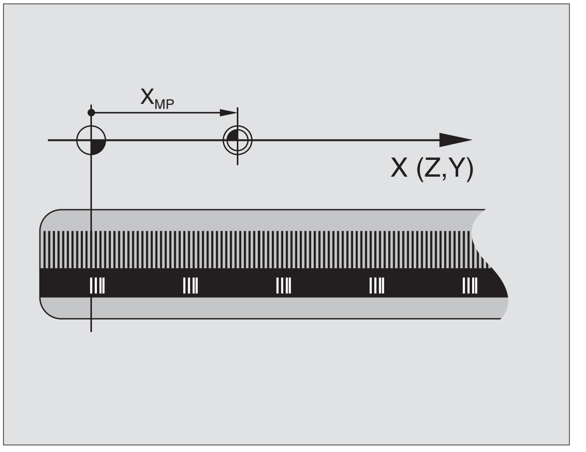
4

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

4.1 Fundamentals

Position encoders and reference marks

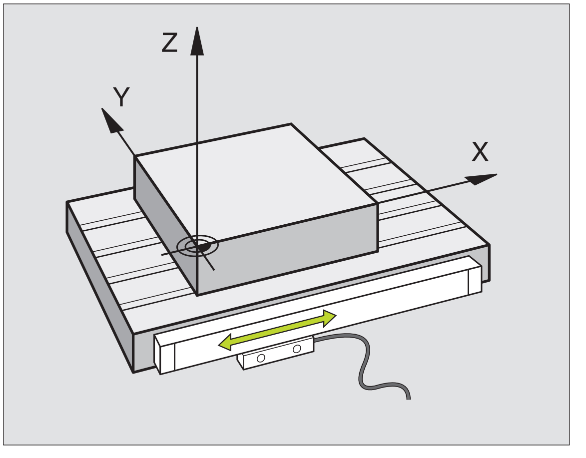
The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders, rotary tables and tilting axes with angle encoders.



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

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

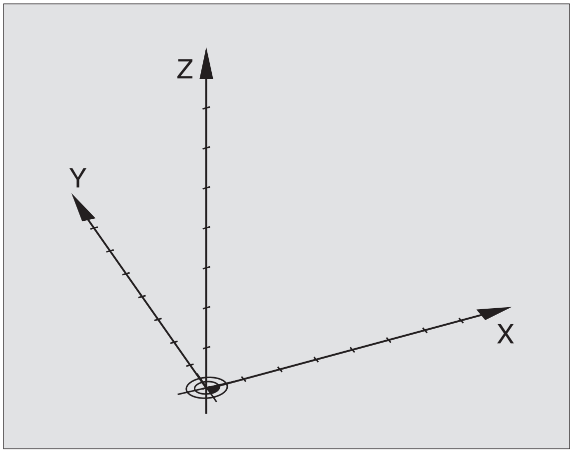


Reference system

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

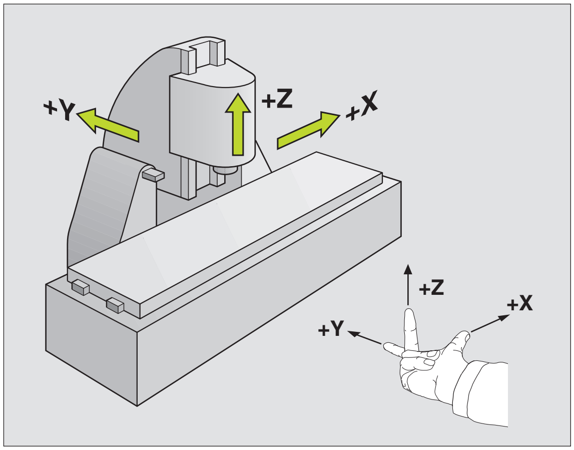
The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is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

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

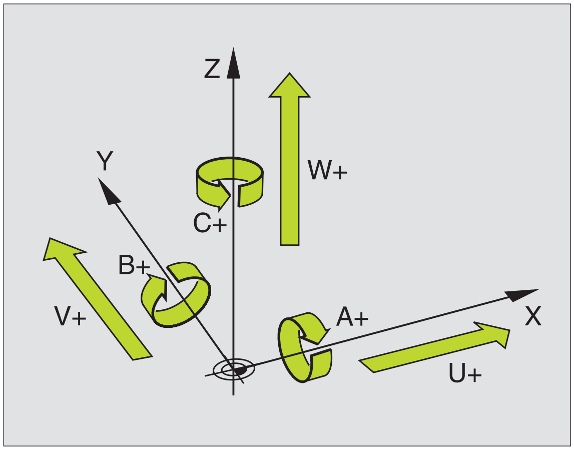


Reference system on milling machines

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



The 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.



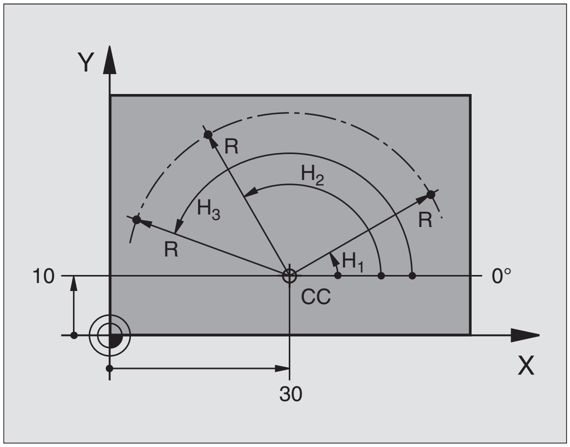
Polar coordinates

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 a circle center (CC), or pole. A position in a plane can be clearly defined by the:

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

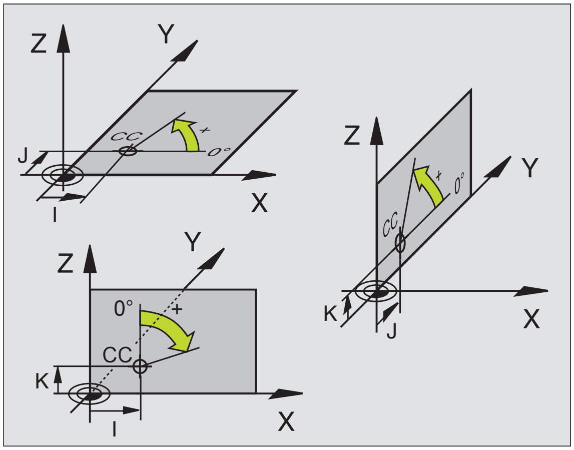
See figure at upper right.



Definition of pole and angle reference axis

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

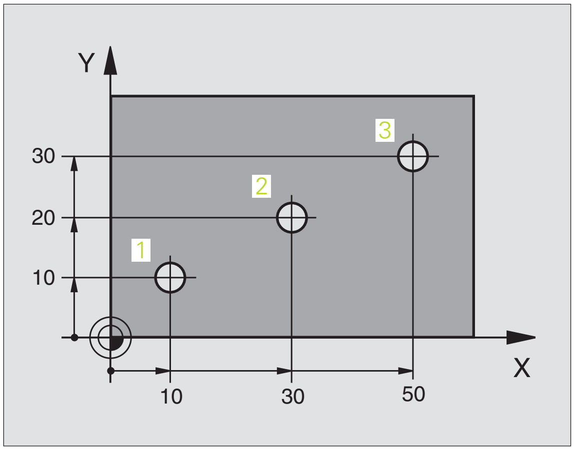
Coordinates of the pole (plane)	Reference axis of the angle
X/Y	+X
Y/Z	+Y
Z/X	+Z



Absolute and incremental workpiece positions

Absolute workpiece positions

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



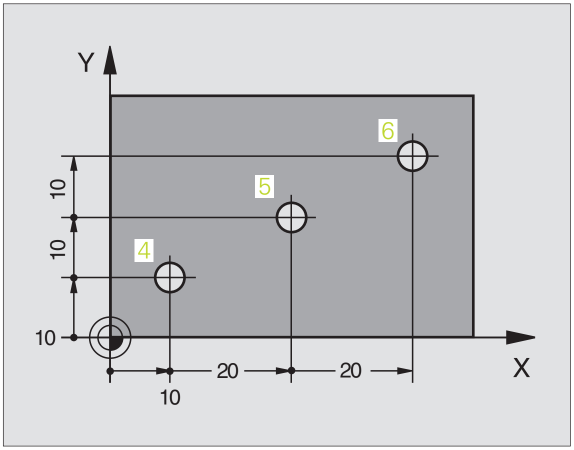
Example 1: Holes dimensioned in absolute coordinates

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

Incremental workpiece positions

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

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**

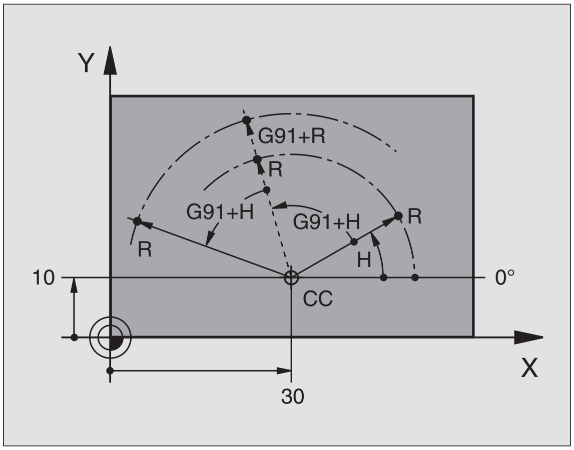
X = 10 mm
Y = 10 mm

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

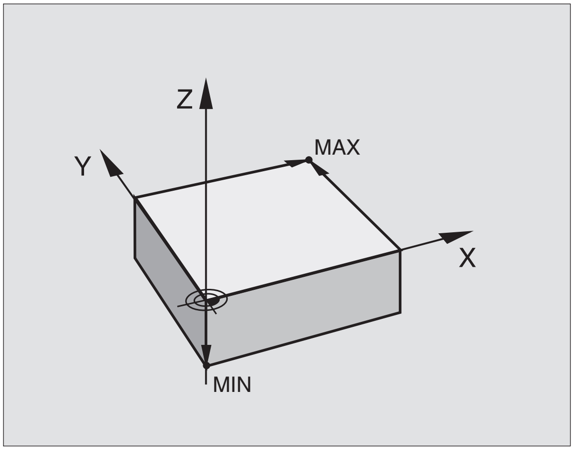
Absolute and incremental polar coordinates

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

Incremental 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. When setting the datum, you first align the workpiece along the machine axes, and then move the tool in each axis to a defined position relative to the workpiece. Set the display of the TNC either to zero or to a known position value for each position. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.

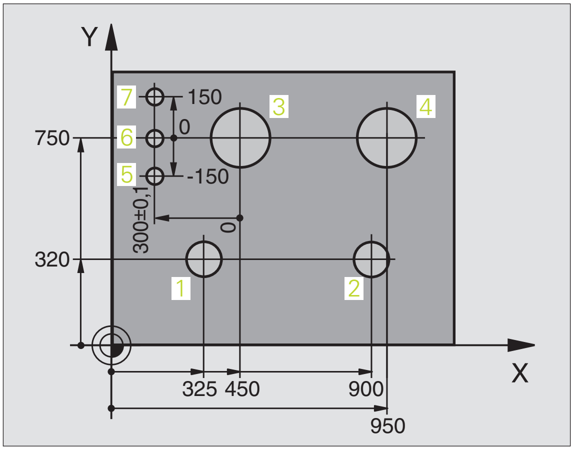
If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles (see "Coordinate Transformation Cycles" on page 438).

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

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

Example

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



4.2 File Management: Fundamentals

Files

Files in the TNC	Type
Programs	
In HEIDENHAIN format	.H
In ISO format	.I
smarT.NC files	
Structured unit program	.HU
Contour descriptions	.HC
Point tables for machining positions	.HP
Tables for	
Tools	.T
Tool changers	.TCH
Pallets	.P
Datums	.D
Points	.PNT
Presets	.PR
Cutting data	.CDT
Cutting materials, workpiece materials	.TAB
Dependent data (such as structure items)	.DEP
Texts as	
ASCII files	.A
Help files	.CHM
Drawing data as	
ASCII files	.DXF

When you write a part program on the TNC, you must first enter a file name. The TNC saves the program to the hard disk 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 nearly any number of files with the TNC, at least **25 GB** (dual-processor version: **13 GB**).

File names

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

PROG20	.I
File name	File type

File names should not exceed 25 characters, otherwise the TNC cannot display the entire file name. The characters *** \ / " ? < > .** are not permitted in file names.

> You cannot use any other special characters, including space characters, in file names.
>
> The maximum limit for the path and file name together is 256 characters (see "Paths" on page 111).

Data backup

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

The TNCremo NT data transmission freeware from HEIDENHAIN is a simple and convenient method for backing up data stored on the TNC.

You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored. Ask your machine manufacturer for assistance, if necessary.

> 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 work hours, e.g. during the night.
>
> Take the time occasionally to delete any unneeded files so that the TNC always has enough hard-disk space for system files (such as the tool table) .

> 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 Working with the File Manager

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 "\".

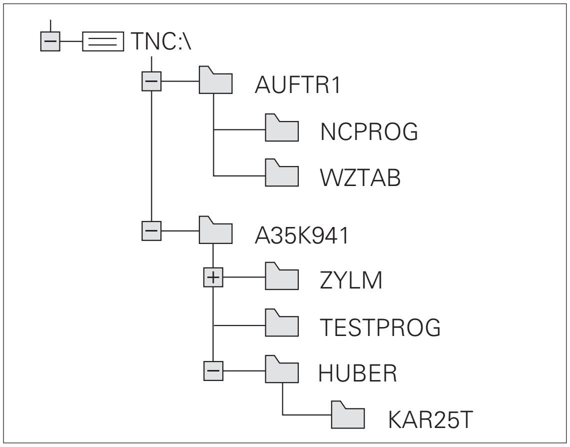
> The path, including all drive characters, directories and the file name, cannot exceed 256 characters!

Example

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

TNC:\AUFTR1\NCPROG\PROG1.H

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

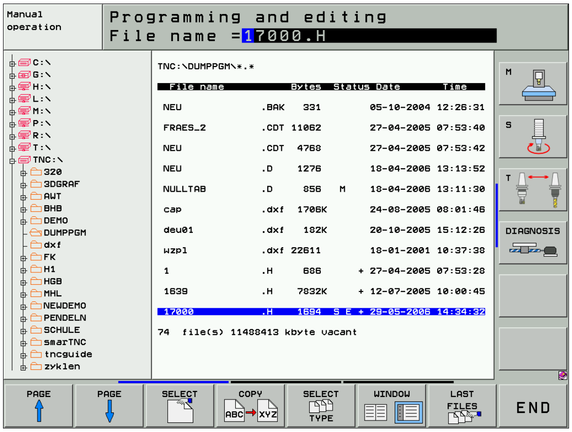


Overview: Functions of the file manager

Function	Soft key	Page
Copy (and convert) individual files	COPY ABC → XYZ	Page 117
Select target directory		Page 117
Display a specific file type	SELECT TYPE	Page 114
Display the last 10 files that were selected	LAST FILES	Page 119
Erase a file or directory	DELETE	Page 120
Mark a file	TAG	Page 121
Rename a file	RENAME ABC = XYZ	Page 122
Protect a file against editing and erasure	PROTECT	Page 122
Cancel file protection	UNPROTECT	Page 122
Manage network drives	NET	Page 126
Copy a directory	COPY DIR	Page 119
Display all the directories of a particular drive	UPDATE TREE	
Delete directory with all its subdirectories	DELETE ALL	Page 122

Calling the file manager

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



The narrow window on the left 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 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.

Step 1: Select drive

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

SELECT — To select a drive, press the SELECT soft key, or

ENT — Press the ENT key.

Step 2: Select a directory

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

Step 3: Select a file

SELECT TYPE — Press the SELECT TYPE soft key.

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

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

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 — Press the SELECT soft key, or

ENT — Press the ENT key.

The TNC opens the selected file in the operating mode from which you called the file manager:

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

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

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

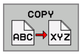
CREATE \NEW DIRECTORY?

YES Press the YES soft key to confirm, or

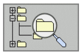
NO Abort with the NO soft key.

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 the target entry" soft key to place a target directory in the 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 to the active directory or to 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.
>
> The file extension of the externally created table should be **.A** (ASCII). In these cases the table can contain any number of lines. If you create a file of type *.T, then the table must contain sequential line numbers beginning with 0.

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.A with 10 lines (for the 10 tools) and the columns

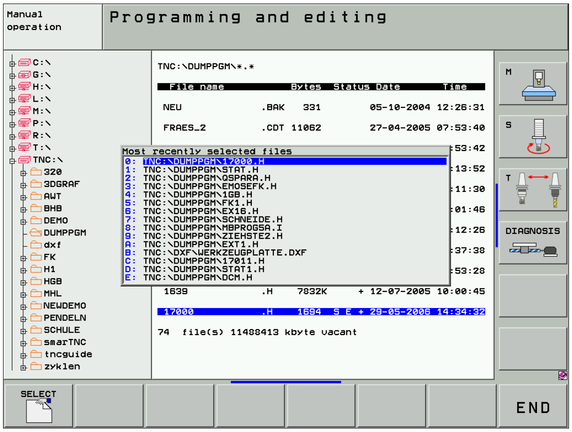
- Tool number (column **T**)
- Tool length (column **L**)
- Tool radius (column **R**)

- ▶ Copy this table from the external date medium to any directory.
- ▶ Copy the externally created table 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 column number, length and radius in the TOOL.T file. The data of the other lines and columns is not changed.
- ▶ Or, if you press the REPLACE EMPTY LINES soft key, the TNC merely overwrites the lines in the TOOL.T file that do not contain any data. The data of the other lines and columns is not changed.

Copying a directory

Move the highlight in the left window onto the directory you want to copy. 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 files selected



PGM MGT — Call the file manager

LAST FILES — Display the last 15 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 — To select a drive, press the SELECT soft key, or

ENT — Press the ENT key.

Deleting a file

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

- To select the erasing function, press the DELETE soft key The TNC asks whether you really want to erase the file.
- To confirm, press the YES soft key, or
- To abort erasure, press the NO soft key.

Deleting a directory

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

- To select the erasing function, press the DELETE soft key The TNC asks whether you really want to erase the directory.
- To confirm, press the YES soft key, or
- To abort erasure, press the NO soft key.

Marking files

Marking functions	Soft key
Mark a single file	TAG FILE
Mark all files in the directory	TAG ALL FILES
Unmark a single file	UNTAG FILE
Unmark all files	UNTAG ALL FILES
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 mark several files, proceed as follows:

Move the highlight to the first file.

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

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

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

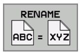
TAG FILE — To mark more files, press the MARK FILE soft key.

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

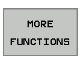
- Move the highlight to the file you want to protect.

- 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

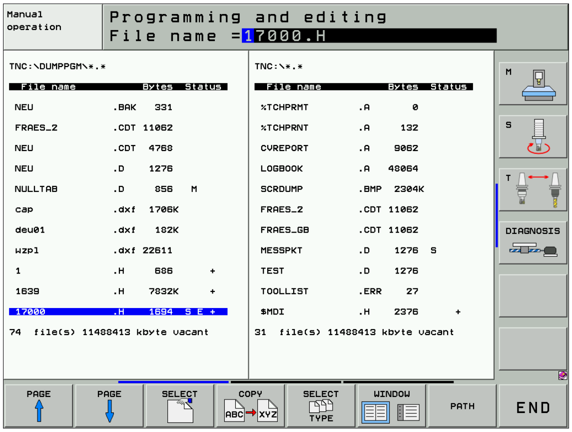
- Move the highlight in the left window onto the directory you want to erase.



- 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 599).

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 the TNC shows all files saved on its hard disk. In the right half of the screen 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.

PATH

To select another drive or directory: Press the PATH soft key. TNC opens a pop-up window. Select the desired directory in the pop-up window by using the arrow keys and the ENT key.

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 "Marking files," page 121)

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.

WINDOW

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

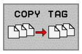
- Move the highlight to the directory into which you wish to copy the files, and display the files in this directory with the ENT key.

In the left window

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

- Display the file marking functions.

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



- Copy the marked files into the target directory.

Additional marking functions: see “Marking files,” page 121.

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

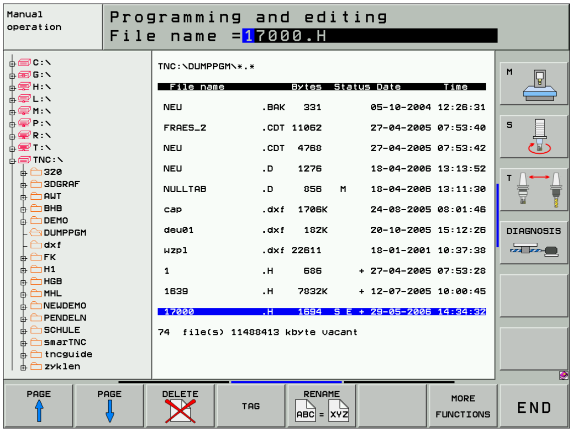
Overwriting files

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," page 603.
>
> To connect the iTNC with Windows 2000 to your network, see "Network Settings," page 661.
>
> The TNC logs error messages during network operation (see "Ethernet Interface" on page 603).

If the TNC is connected to a network, the directory window displays up to 7 drives (see figure). 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 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]**. The maximum transmission speed is 2 to 5 MB/s, depending on the type of file being transferred and how busy the network is.

USB devices on the TNC (FCL 2 function)

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

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

The TNC automatically detects these types of USB devices when connected. The TNC does not support USB devices with other file systems (such as NTFS). The TNC displays the **USB: TNC does not support device** error message when such a device is connected.

> The TNC also displays the **USB: TNC does not support device** error message if you connect a USB hub. In this case simply acknowledge the message with the CE key.
>
> In theory, you should be able to connect all USB devices with the file systems mentioned above to the TNC. If problems occur nevertheless, please contact HEIDENHAIN.

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

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

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

▶ Select the left window with the arrow key.

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

▶ Scroll through the soft-key row.

NET ▶ Select additional functions.

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

END ▶ Exit program management.

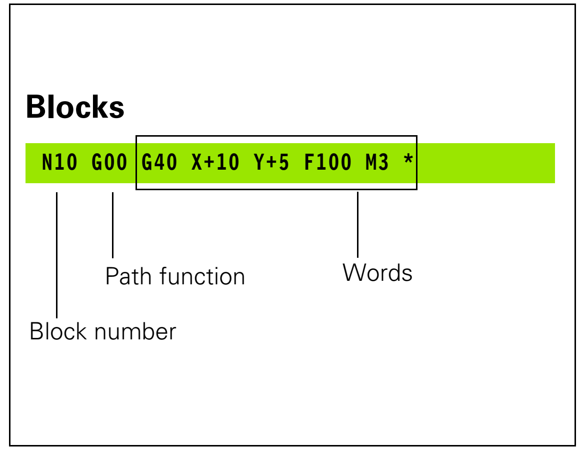
In order to re-establish a connection with a USB device that has been removed, press the following soft key:

▶ Select the function for reconnection of USB devices.

4.4 Creating and Writing Programs

Organization of an NC program in 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 calls
- Feed rates and spindle speeds, as well as
- Approaching a safe position
- 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).

> After each tool call, HEIDENHAIN recommends always traversing to a safe position, from which the TNC can position the tool for machining without causing a collision!

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. An example of program initiation:

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

PGM MGT

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

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

FILE NAME = 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 **BLK FORM** (workpiece blank).

WORKING SPINDLE AXIS X/Y/Z ?

Z

Enter the spindle axis.

DEF BLK FORM: MIN-CORNER ?

0	ENT	Enter in sequence the X, Y and Z coordinates of the MIN point.
0	ENT	
-40	ENT	

DEF BLK FORM: MAX-CORNER ?

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

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

You cannot edit a program while it is being run by the TNC in a machine operating mode. The TNC allows you to place the cursor in the block, but it does not save the changes and responds instead with an error message.

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

Function	Soft key/key
Go to previous page	PAGE ↑
Go to next page	PAGE ↓
Go to beginning of program	BEGIN ↑
Go to end of program	END ↓
Change the position of the current block on the screen: Press this soft key to display additional program blocks that are programmed before the current block.	
Change the position of the current block on the screen: Press this soft key to display additional program blocks that are programmed after the current block.	
Move from one block to the next	↓ ↑
Select individual words in a block	← →
To select a certain block, press the GOTO key, enter the desired block number, and confirm with the ENT key. Or: Enter the block number step and press the N LINES soft key to jump over the entered number of lines upward or downward.	GOTO

Function	Soft key/key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
Delete the selected block	DEL
Erase cycles and program sections	DEL
Insert the block that you 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

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

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

Select a block with the arrow keys.

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

If you have started a search in a very long program, the TNC shows a progress display window. You then have the option of canceling the search via soft key.

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

Finding any text

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

Marking, copying, deleting and inserting program sections

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

To copy a program section, proceed as follows:

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

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

- To insert the block, press the INSERT BLOCK soft key.

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

The TNC search function

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

Searching for texts

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

FIND

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

G +40

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

CONTINUE

- Start the search process: The TNC displays the available search options in the soft-key row (see 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 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 continues until it reaches it again.	COMPLETE/BEGIN/END; COMPLETE/BEGIN/END
Start a new search.	NEW SEARCH

Find/Replace any text

The find/replace function is not possible if

- a program is protected
- the program is currently being run by the TNC

When using the REPLACE ALL function, ensure that you do not accidentally replace text that you do not want to change. Once replaced, such text cannot be restored.

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

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

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

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

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

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

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

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

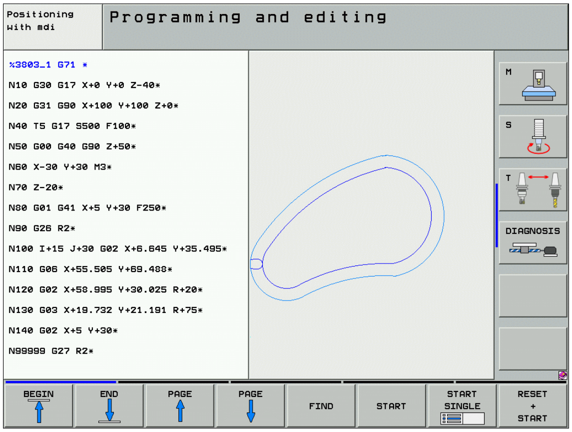
EXECUTE
- To replace the text and then move to the next occurrence of the text, press the REPLACE soft key. To replace all text occurrences, press the REPLACE soft key. To skip the text and move to its next occurrence press the DO NOT REPLACE soft key.

END
- End the search function.

4.5 Interactive Programming Graphics

Generating / Not generating graphics during programming:

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



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

AUTO DRAW OFF ON

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

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

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

Generating a graphic for an existing program

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

RESET + START

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

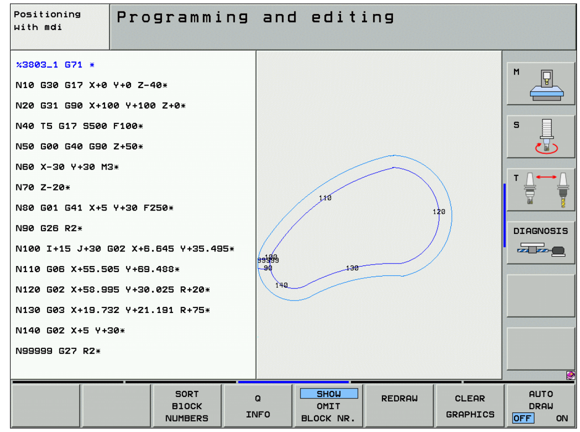
Additional functions:

Function	Soft key
Generate a complete graphic	RESET + START
Generate interactive graphic blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Stop the programming graphics. This soft key only appears while the TNC is generating the interactive graphics	STOP
Redraw the programming graphics, for example if lines were deleted by intersections	REDRAW

Block number display ON/OFF

SHOW OMIT BLOCK NR.

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



Erase the graphic

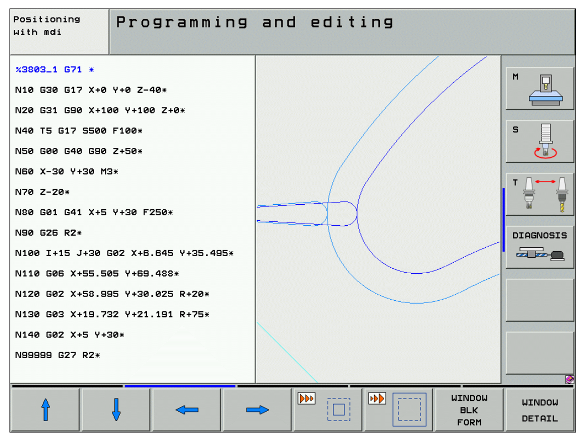
CLEAR GRAPHICS

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

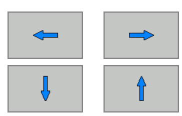
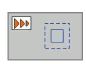
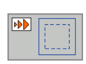
Magnifying or reducing a detail

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

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



The following functions are available:

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

WINDOW DETAIL

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

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

4.6 3-D Line Graphics (FCL 2 Function)

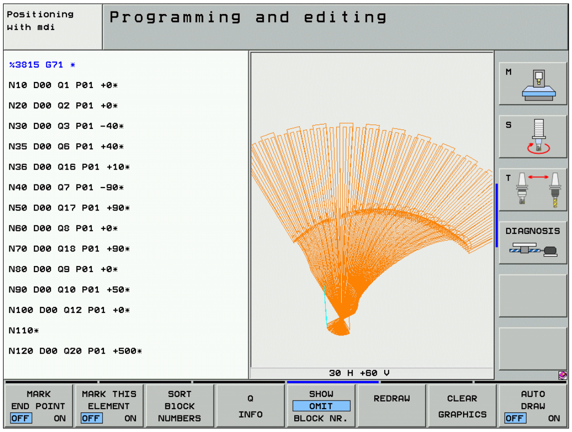
Function

Use the 3-D line graphics to have the TNC show the programmed traverse paths in three dimensions. A powerful zoom function is available for recognizing details quickly.

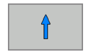
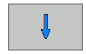
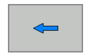
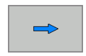
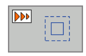
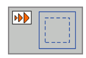
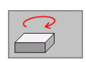
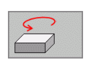
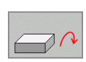
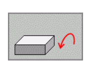
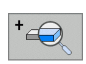
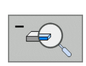
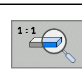
You should especially use the 3-D line graphics to inspect programs created externally for irregularities before machining, in order to avoid undesirable traces of the machining process on the workpiece. Such traces of machining can occur when points are output incorrectly by the postprocessor.

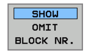
In order to find the error location quickly, the TNC shows the currently active block of the 3-D line graphics in a different color in the left window (default setting: red).

- To switch the screen layout to displaying program blocks to the left and 3-D line graphics to the right, press the SPLIT SCREEN key and PROGRAM + 3D LINES soft key.



Functions of the 3-D line graphics

Function	Soft key
Show and move the zoom frame up. Press and hold the soft key to move the frame.	
Show and move the zoom frame down. Press and hold the soft key to move the frame.	
Show and move the zoom frame to the left. Press and hold the soft key to move the frame.	
Show and move the zoom to the right. Press and hold the soft key to move the frame.	
Enlarge the frame overlay—press and hold the soft key to magnify the detail.	
Reduce the frame overlay—press and hold the soft key to reduce the detail.	
Reset detail magnification so that the workpiece is displayed as it was programmed with BLK FORM.	WINDOW BLK FORM
Select the isolated detail	TRANSFER DETAIL
Rotate workpiece clockwise	
Rotate workpiece counterclockwise	
Tilt workpiece backward	
Tilt workpiece forward	
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 reduced, the TNC shows the letter **Z** in the footer of the graphic window.	
Show workpiece at original size	
Show workpiece in the last active view	LAST VIEW
Show/hide programmed end points with a dot on the line	MARK END POINT OFF ON

Function	Soft key
Do or do not highlight the selected NC block of the 3-D line graphics in the left window	
Do or do not show block numbers	

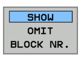
You can also use the mouse with the 3-D line graphics. The following functions are available:

- In order to rotate the wire model shown in three dimensions: Hold the right mouse button down and move the mouse. The TNC displays a coordinate system showing the currently active orientation of the workpiece. After you release the right mouse button, the TNC orients the workpiece to the defined orientation.
- In order to shift the wire model shown: Hold the center mouse button or the wheel button down and move the mouse. The TNC shifts the workpiece in the corresponding direction. After you release the center mouse button, the TNC shifts the workpiece to the defined position.
- In order to zoom in on a certain area with the mouse: Draw a rectangular zoom area while holding the left mouse button down. After you release the left mouse button, the TNC zooms in on the defined area of the workpiece.
- In order to quickly zoom in and out with the mouse: Rotate the wheel button forward or backward.

Highlighting NC blocks in the graphics


- Shift the soft-key row.
- To highlight the NC block selected in the left window in the 3-D line graphics in the right window, set the MARK THIS ELEMENT OFF / ON soft key to ON.
- To not highlight the NC block selected in the left window in the 3-D line graphics in the right window, set the MARK THIS ELEMENT OFF / ON soft key to OFF.

Block number display ON/OFF

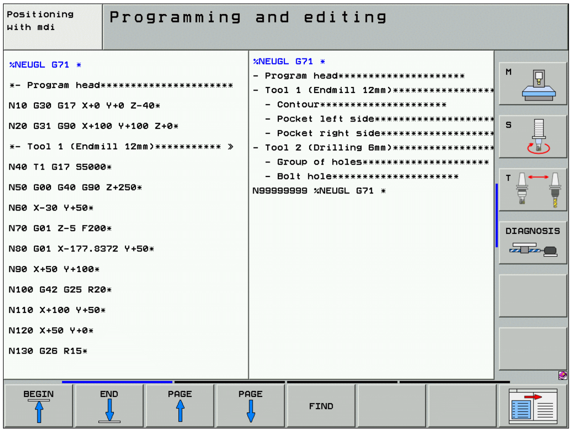
- Shift the soft-key row.
- To show block numbers: Set the SHOW OMIT BLOCK NR. soft key to SHOW.
- To omit block numbers: Set the SHOW OMIT BLOCK NR. soft key to OMIT.

Erase the graphic


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

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 37 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.

The inserted structure items are managed by the TNC in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

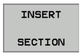
Displaying the program structure window / Changing the active window

- 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.

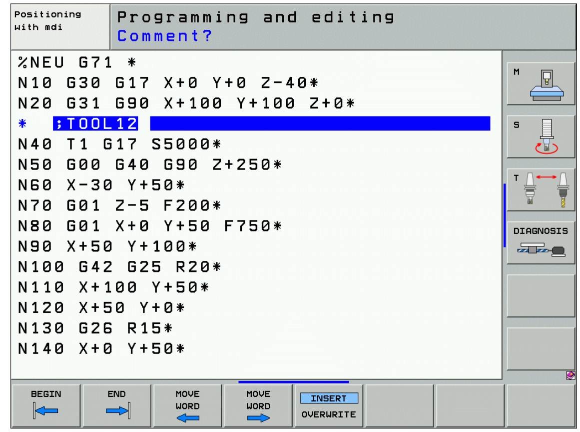
Selecting blocks in the program structure window

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.

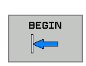
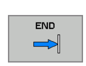
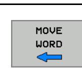
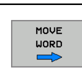
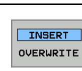
Inserting comments after program entry

- Select the block to which a comment is to be added.
- Select the last word in the block with the right arrow key: A semicolon appears at the end of the block and 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.	

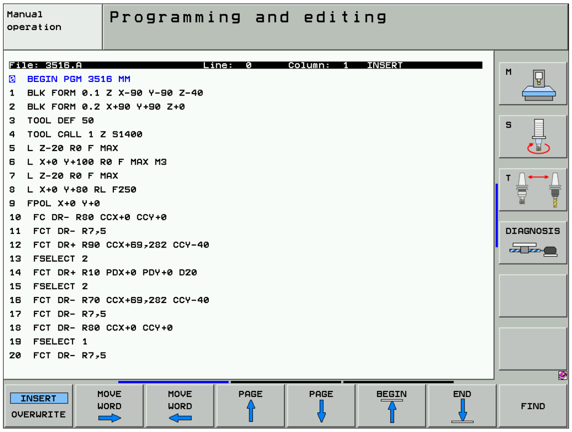
4.9 Creating Text Files

Function

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

- Recording test results
- Documenting working procedures
- Creating formularies

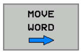
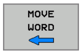
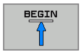
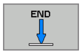
Text files are type .A files (ASCII files). If you want to edit other types of files, you must first convert them into type .A files.



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 one word to the left	
Go to next screen page	
Go to previous screen page	
Go to beginning of file	
Go to end of file	

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

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.

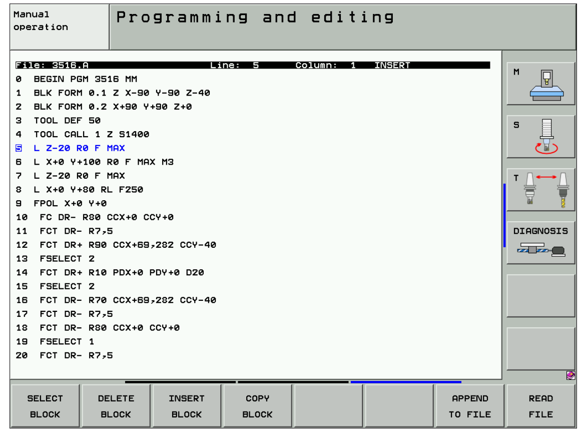
Deleting and inserting characters, words and lines

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 to 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 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 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.

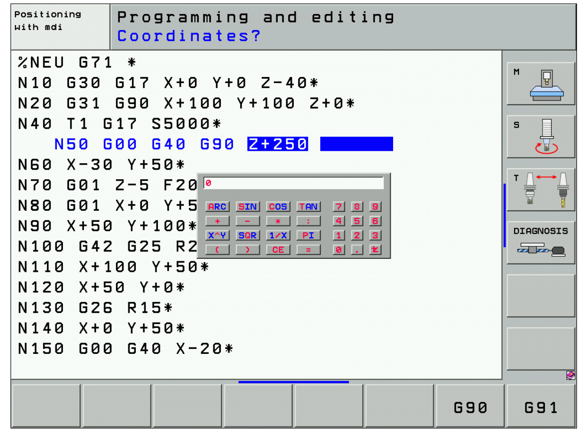
Finding any text

- 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	( )
pi (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

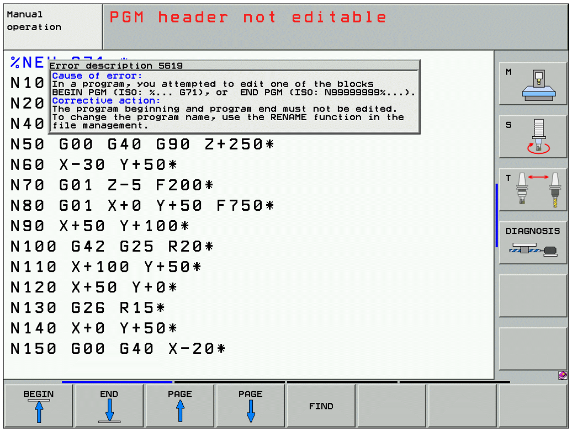
An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. The TNC error messages can be canceled with the CE key, after the cause of the error has been removed.

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 cause of error and any suggestions for possible remedies. The TNC may show additional information that can be helpful to trained HEIDENHAIN personnel during troubleshooting. 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.

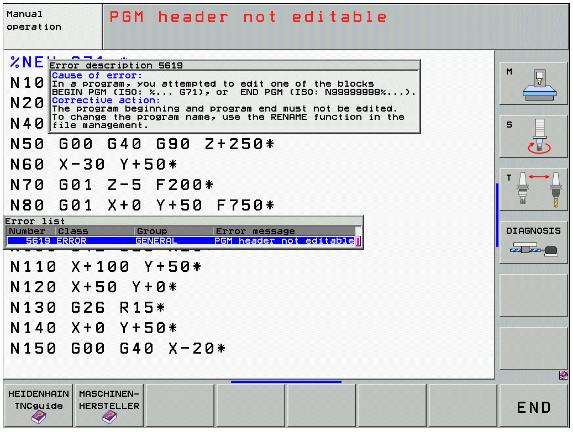


The TNC displays the Help text automatically if the error message is blinking. The TNC needs to be restarted after blinking error messages. To restart the TNC, press and hold the END key for two seconds.

4.12 List of All Current Error Messages

Function

With this function you can show a pop-up window in which the TNC shows all current error messages. The TNC shows errors both from the NC as well as those from the machine tool builder.



Show error list

You can call the list as soon as at least one error message is present:

ERR

- To display the list, press the ERR key.
- You can select one of the current error messages with the arrow keys.
- With the CE key or the DEL key you can delete the error message from the pop-up window momentarily selected. When you delete the last error message, the pop-up window closes as well.
- To close the pop-up window, press the ERR key again. Current error messages are retained.

Parallel to the error list you can also view the respective help text in a separate window: Press the HELP key.

Calling the TNCguide help system

You can call the TNC's help system via soft key. Immediately the help system shows you the same error explanation that you receive by pressing the HELP soft key.

If your machine manufacturer also provides a help system, the TNC shows an additional MACHINE MANUFACTURER soft key with which you can call this separate help system. There you will find further, more detailed information on the error message concerned.

HEIDENHAIN TNCguide

- Call the help for HEIDENHAIN error messages.

MACHINE MFR.

- Call the help for HEIDENHAIN error messages, if available.

Window contents

Column	Meaning
`Number`	Error number (–1: no error number defined), issued by HEIDENHAIN or your machine tool builder
`Class`	Error class. Defines how the TNC processes this error. ■ **ERROR** Program run is interrupted by the TNC (INTERNAL STOP) ■ **FEED HOLD** The feed-rate release is deleted ■ **PGM HOLD** The program run is interrupted (the control-in-operation symbol blinks) ■ **PGM ABORT** The program run is interrupted (INTERNAL STOP) ■ **EMERG. STOP** EMERGENCY STOP is set off ■ **RESET** TNC executes a system restart ■ **WARNING** Warning message, the program run resumes ■ **INFO** Info message, the program run resumes
`Group`	Group. Specifies from which section of the operating system software the error message was generated: ■ **OPERATING** ■ **PROGRAMMING** ■ **PLC** ■ **GENERAL**
`Error message`	Respective error text displayed by the TNC

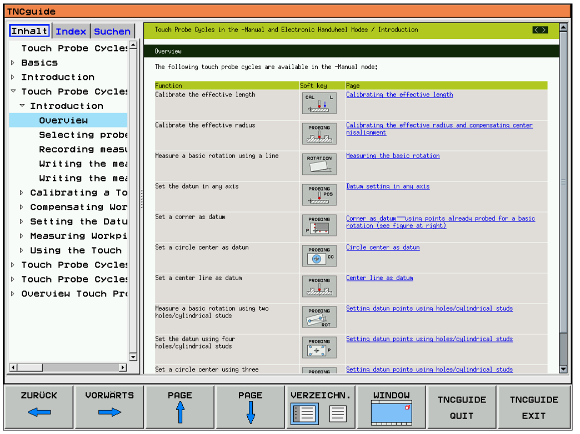
4.13 The Context-Sensitive Help System TNCguide (FCL3 Function)

Function

The TNCguide help system is only available if your control hardware has as least 256 MB RAM and the FCL3 is enabled.

The **TNCguide** context-sensitive help system includes the user documentation in HTML format. The TNCguide is called with the HELP key, and the TNC often immediately displays the information specific to the condition from which the help was called (context-sensitive call).

The English and German documentation is shipped as standard with each NC software level. HEIDENHAIN provides the remaining conversational languages for cost-free download as soon as the respective translations are available (see “Downloading current help files” on page 161).



The TNC always tries to start the TNCguide in the language that you have selected as the conversational language on your TNC. If the files with this language are not yet available on your TNC, it automatically opens the English version.

The following user documentation is currently available in the TNCguide:

- Conversational Programming User’s Manual **(BHBKlartext.chm)**
- Touch Probe Cycles User’s Manual **(BHBtchprobe.chm)**
- User’s Manual for smarT.NC **(BHBSmart.chm)** (same format as a “Pilot”)
- List of All Error Messages **(errors.chm)**

In addition, the **main.chm** “book” file is available containing the content of all existing .chm files together.

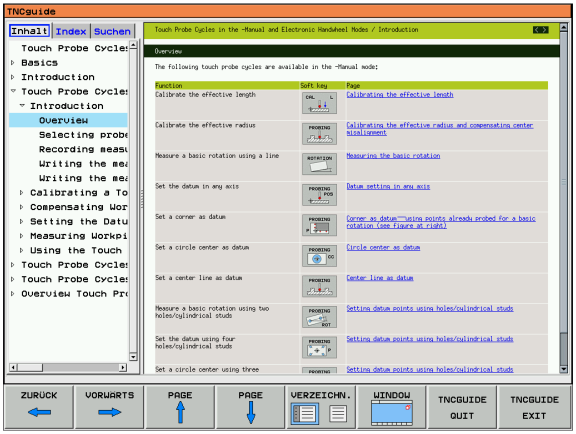
As an option, your machine tool builder can embed machine-specific documentation in the **TNCguide.** These documents then appear as a separate book in the **main.chm** file.

Working with the TNCguide

Calling the TNCguide

There are several ways to start TNCguide:

- Press the HELP key if the TNC is not already showing an error message.
- Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys.
- Use the file manager to open a help file (chm file). The TNC can open any .chm file, even if it isn't saved on the TNC's hard disk.



> If one or more error messages are waiting for your attention, the TNC shows the help directly associate with the error messages. To start the **TNCguide,** you first have to acknowledge all error messages.
>
> When the help system is called on the programming station or the dual-processor version, the TNC starts the internally define standard browser (usually the Internet Explorer), and on the single-processor version browser adapted by HEIDENHAIN.

For many soft keys there is a context-sensitive call through which you can go directly to the description of the soft key's function. This functionality requires using a mouse. Proceed as follows:

- Select the soft-key row containing the desired soft key.
- Click with the mouse on the help symbol that the TNC displays just above the soft-key row: The mouse pointer turns into a question mark.
- Move the question mark to the soft key for which you want an explanation, and click: The TNC opens the TNCguide (conversational dialog documentation). If no specific part of the help is assigned to the selected soft key, the TNC opens the book file **main.chm,** in which you can use the search function or the navigation to find the desired explanation manually.

Navigating in the TNCguide

It’s easiest to use the mouse to navigate in the TNCguide. A table of contents appears on the left side of the screen. By clicking the rightwardly pointing triangle you open subordinate sections, and by clicking the respective entry you open the individual pages. It is operated in the same manner as the Windows Explorer.

Linked text positions (cross references) are shown underlined and in blue. Clicking the link opens the associated page.

Of course you can also operate the TNCguide through keys and soft keys. The following table contains on overview of the corresponding key functions.

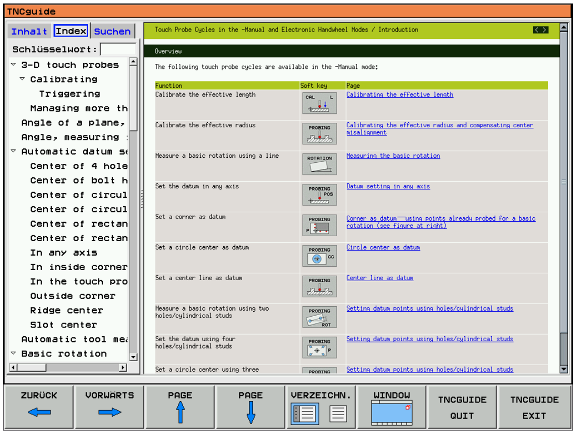
> The key functions described below are only available on the single-processor version of the TNC.

Function	Soft key
■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely.	 
■ If the table of contents at left is active: Open a branch of the table of contents. If the branch is at its end, jump into the window at right. ■ If the text window at right is active: No function	
■ If the table of contents at left is active: Close a branch of the table of contents. ■ If the text window at right is active: No function	
■ If the table of contents at left is active: Use the cursor key to show the selected page ■ If the text window at right is active: If the cursor is on a link, jump to the linked page	
■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the screen half at right. ■ If the text window at right is active: Jump back to the window at left	
■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Jump to the next link	 

Function	Soft key
Select the page last shown	BACK ←
Page forward if you have used the "select page last shown" function	FORWARD →
Move up by one page	PAGE ↑
Move down by one page	PAGE ↓
Display or hide table of contents	DIRECTORY
Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the TNC window.	WINDOW
The focus is switched internally to the TNC application so that you can operate the control when the TNCguide is open. If the full screen is active, the TNC reduces the window size automatically before the change of focus.	TNCGUIDE QUIT
Close the TNCguide	TNCGUIDE EXIT

Subject index

The most important subjects in the Manual are listed in the subject index (**Index** tab). You can select them directly by mouse or with the cursor keys.



The left side is active.

- Select the **Index** tab
- Activate the **Keyword** input field
- Enter the word for the desired subject and the TNC synchronizes the index and creates a list in which you can find the subject more easily, or
- Use the arrow key to highlight the desired keyword
- Use the ENT to call the information on the selected keyword

Full-text search

In the **Find** tab you can search the entire TNCguide for a specific word.

The left side is active.

- Select the **Find** tab
- Activate the **Find:** input field
- Enter the desired word and confirm with the ENT key: The TNC lists all sources containing the word
- Use the arrow key to highlight the desired source
- Press the ENT key to go to the selected source

The full-text search only works for single words.

If you activate the **Search only in titles** function (by mouse or by using the cursor and the space key), the TNC searches only through headings and ignores the body text.

Downloading current help files

You'll find the help files for your TNC software on the HEIDENHAIN Homepage **www.heidenhain.de** under:

- Services and Documentation
- Software
- iTNC 530 help system
- NC software number of your TNC, for example **34049x-03**
- Select the desired language, for example English: You will see a ZIP file with the appropriate help files
- Download the ZIP file and unzip it
- Move the unzipped CHM files to the TNC in the **TNC:\tncguide\en** directory or into the appropriate language subdirectory (see also the following table)

If you want to use TNCremoNT to transfer the CHM files to the TNC, then in the **Extras**>**Configuration**>**Mode**>**Transfer in binary format** menu item you have to enter the **.CHM** extension.

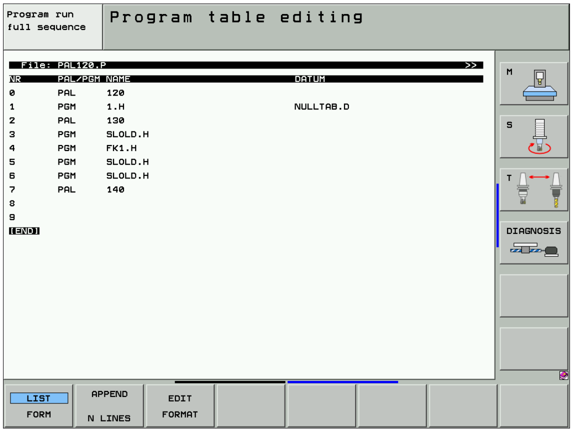
Language	TNC directory
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**
Chinese (traditional)	**TNC:\tncguide\zh-tw**
Slovenian (software option)	**TNC:\tncguide\sl**

Language	TNC directory
Norwegian	`TNC:\tncguide\no`
Slovak	`TNC:\tncguide\sk`
Latvian	`TNC:\tncguide\lv`
Korean	`TNC:\tncguide\kr`
Estonian	`TNC:\tncguide\et`

4.14 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 reference points.

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.
- **PRESET** (entry optional):
  Preset number from the preset table. The preset number defined here is interpreted by the TNC either as a pallet datum **PAL** in the **PAL/PGM**) column or as a workpiece datum (**PGM** entry in the **PAL/PGM**) line.
- **DATUM** (entry optional):
  Name of the datum table. The datum table must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the datum table. Datums from the datum table can be activated in the NC program with Cycle 7 **DATUM SHIFT.**

- **X, Y, Z** (entry optional, other axes also possible):
  For pallet names, the programmed coordinates are referenced to the machine datum. For NC programs, the programmed coordinates are referenced to the pallet datum. These entries overwrite the datum that you last set in the Manual mode of operation. With the miscellaneous function M104 you can reactivate the datum that was last set. With the actual-position-capture key, the TNC opens a window that enables you to have the TNC enter various points as datums (see 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

Editing function	Soft key
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.

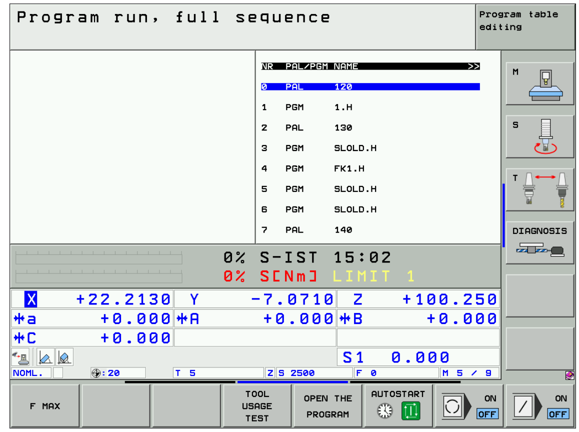
Leaving the pallet file

- 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

MP7683 defines whether the pallet table is to be executed blockwise or continuously .

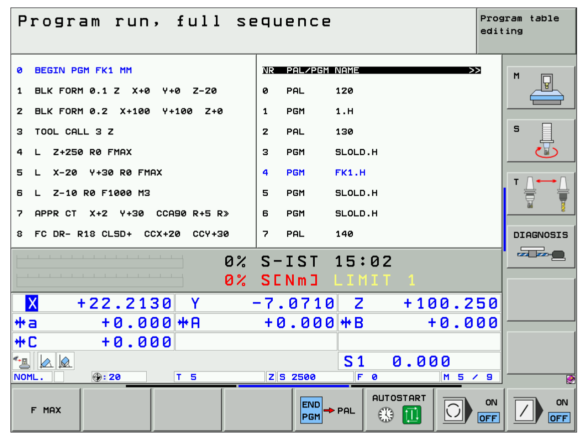
Provided that Machine Parameter 7246 is set so that the tool usage test is active, you can monitor the tool service life for all tools used in a pallet (see "Tool usage test" on page 571).



- 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:

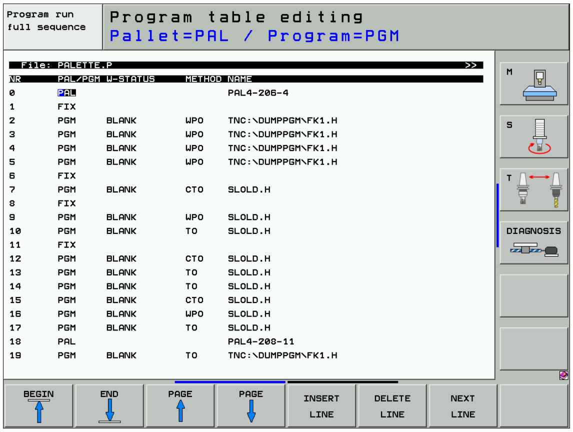


- Select a pallet table.
- With the arrow keys, choose the program you would like to check.
- Press the OPEN PGM soft key: The TNC displays the selected program on the screen. You can now page through the program with the arrow keys.
- To return to the pallet table, press the END PGM soft key.

4.15 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 reference points.

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.
- **PRESET** (entry optional):
  Preset number from the preset table. The preset number defined here is interpreted by the TNC either as a pallet datum **PAL** in the **PAL/PGM**) column or as a workpiece datum (**PGM** entry in the **PAL/PGM**) line.
- **DATUM** (entry optional):
  Name of the datum table. The datum table must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the datum table. Datums from the datum table can be activated in the NC program with Cycle 7 **DATUM SHIFT.**

- **X, Y, Z** (entry optional, other axes also possible):
  For 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
Edit the table format	EDIT FORMAT

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 ↑

Editing function in entry-form mode	Soft key
Select next workpiece	WORKPIECE
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
Delete buffer memory contents	ERASE INTERMED. MEMORY
Tool-optimized machining	TOOL ORIENTAT.
Workpiece-optimized machining	WORKPIECE ORIENTAT.

Editing function in entry-form mode	Soft key
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

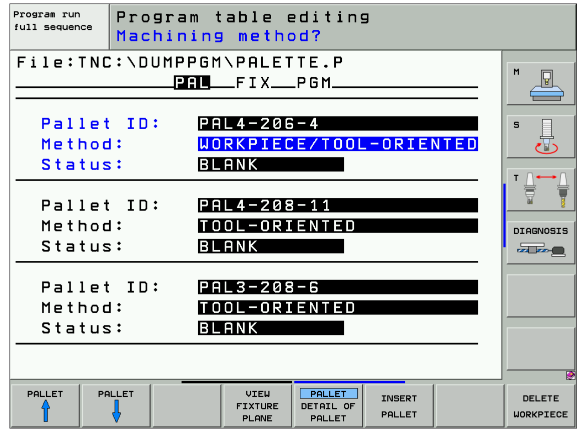
- Call the file manager in the Programming and Editing or Program Run mode: Press the PGM MGT key.
- To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- Select a pallet table with the arrow keys, or enter a new file name to create a new table.
- Confirm your entry with the ENT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

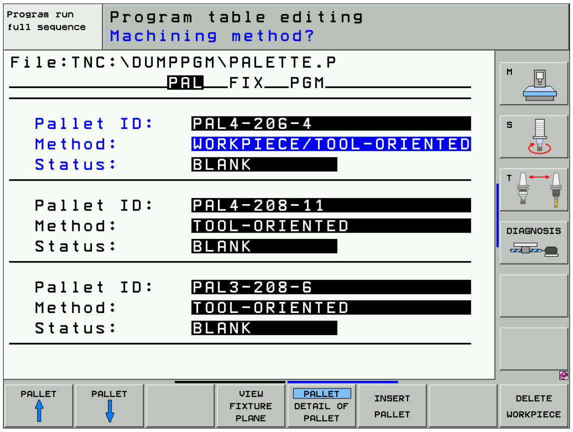


> Set up the pallet file according to the machine configuration. If you only have one fixture with multiple workpieces, then defining one fixture **FIX** with the workpieces **PGM** is sufficient. However, if one pallet contains several fixtures, or if a fixture is machined from more than one side, you 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

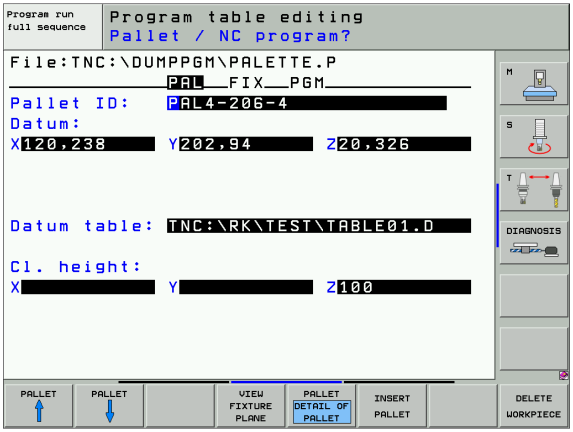
- **`Pallet Id:`** The pallet name is displayed
- **`Method:`** You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. The selected method is assumed for the workpiece level and overwrites any existing entries. In tabular view, WORKPIECE ORIENTED appears as **`WPO`**, and 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 **`EMPTY POSITION`** soft key if you want to skip the pallet during machining. **`EMPTY`** appears in the Status field.

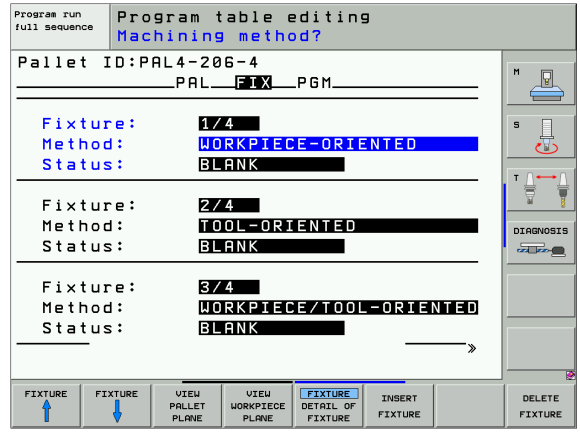
Setting up details in the pallet level



- **`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.

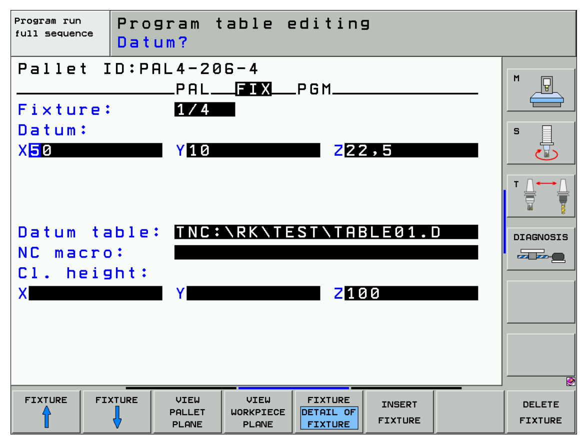


> The TO-/WP-ORIENTED entry cannot be made via soft key. It only appears when different machining methods were chosen for the workpieces in the workpiece level.
>
> If the machining method was determined in the fixture level, the entries are transferred to the workpiece level, where they overwrite any 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 **`EMPTY POSITION`** soft key if you want to skip the fixture during machining. **`EMPTY`** appears in the Status field.

Setting up details in the fixture level

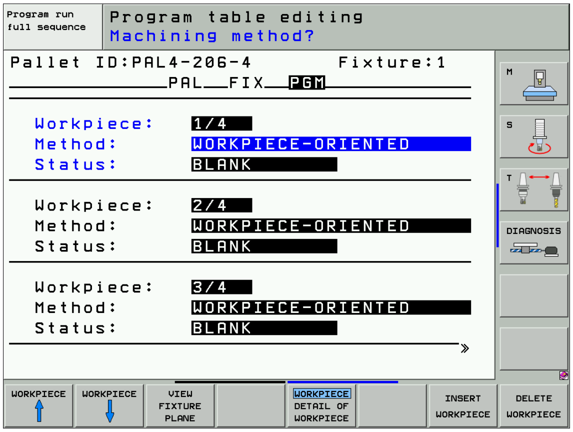
- **`Fixture:`** The number of the fixture is displayed. The number of fixtures within this level is shown after the slash.
- **`Datum:`** Enter the fixture datum.
- **`Datum table:`** Enter the name and path of the datum table valid for machining the workpiece. The data is transferred to the workpiece 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

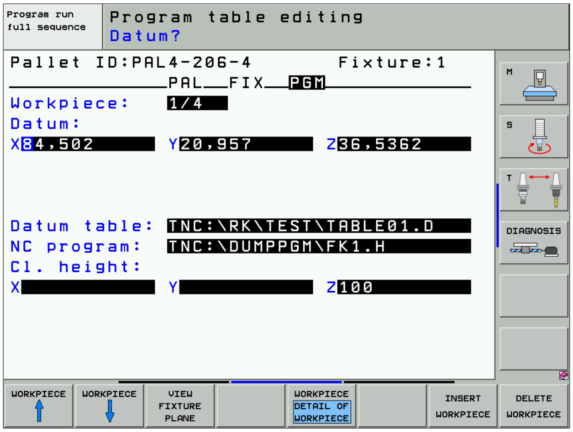
- **`Workpiece:`** The number of the workpiece is displayed. The number of workpieces within this fixture level is shown after the slash.
- **`Method:`** You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. In tabular view, WORKPIECE ORIENTED appears as **`WPO,`** and TOOL 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 2 weeks. 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

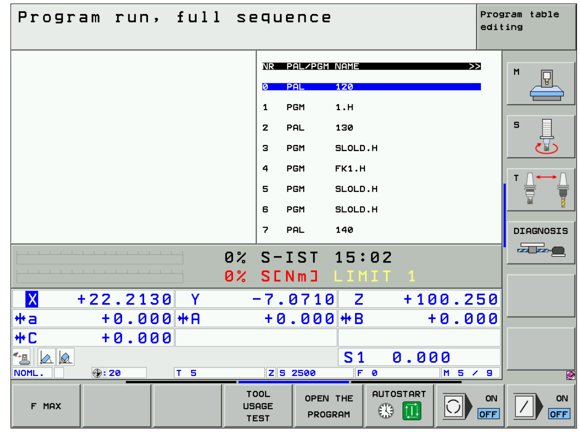
> In MP7683, set whether the pallet table is to be executed blockwise or continuously (see “General User Parameters” on page 628).
>
> Provided that Machine Parameter 7246 is set so that the tool usage test is active, you can monitor the tool service life for all tools used in a pallet (see “Tool usage test” on page 571).

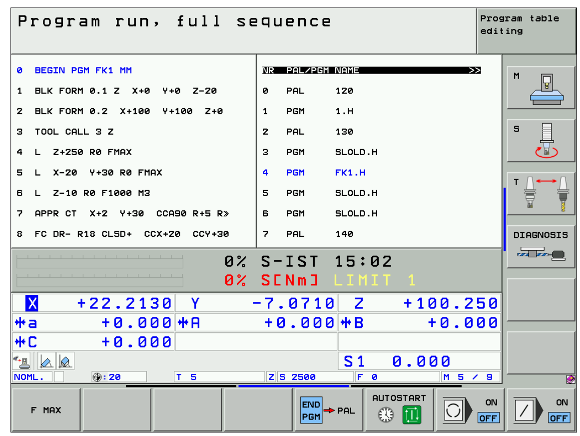
- Select the file manager in the Program Run, Full Sequence or Program Run, Single Block operating modes: Press the PGM MGT key.
- To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- Select the pallet table with the arrow keys and confirm with ENT.
- To execute the pallet table: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

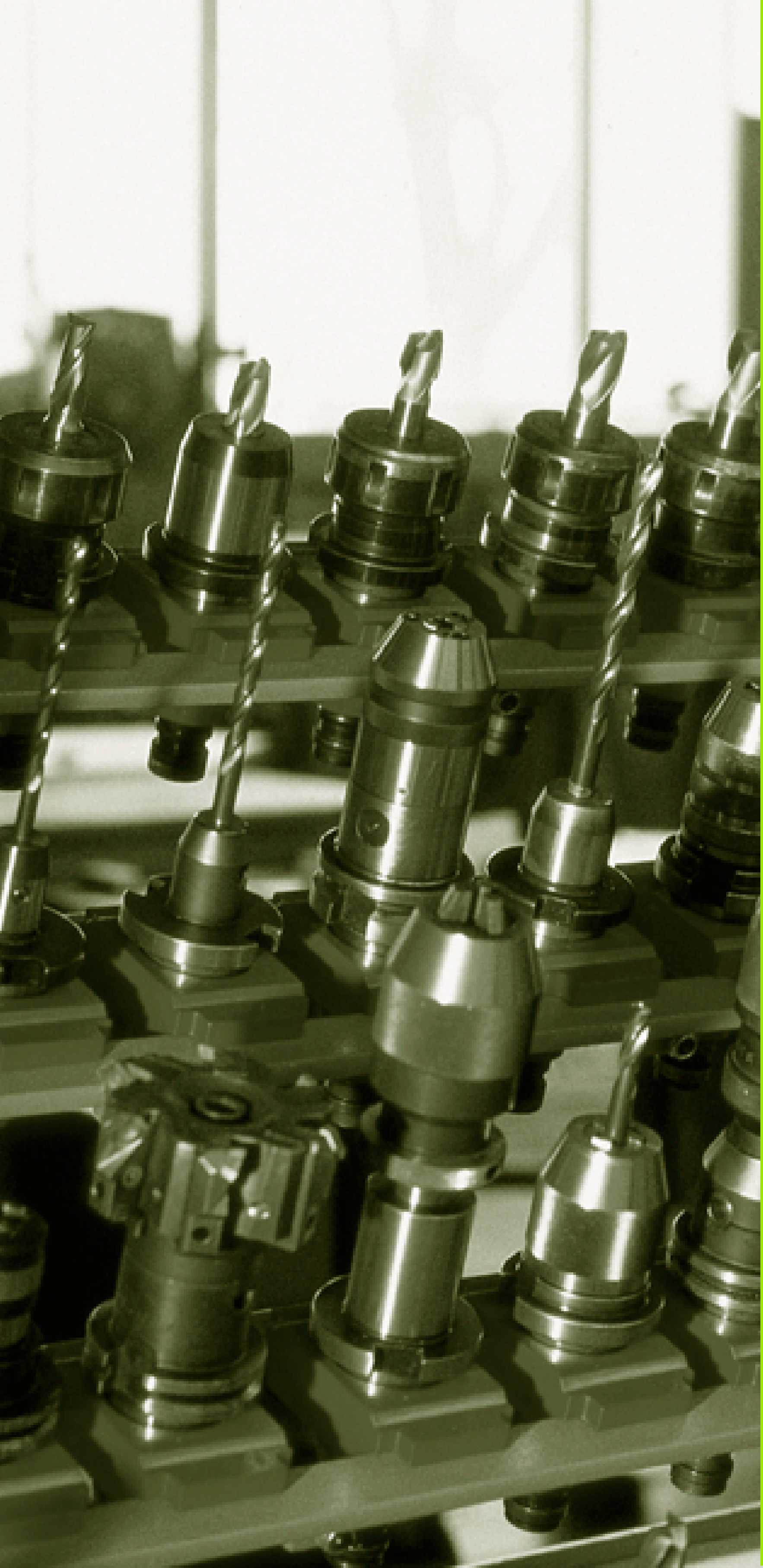
Screen layout for executing pallet tables

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.





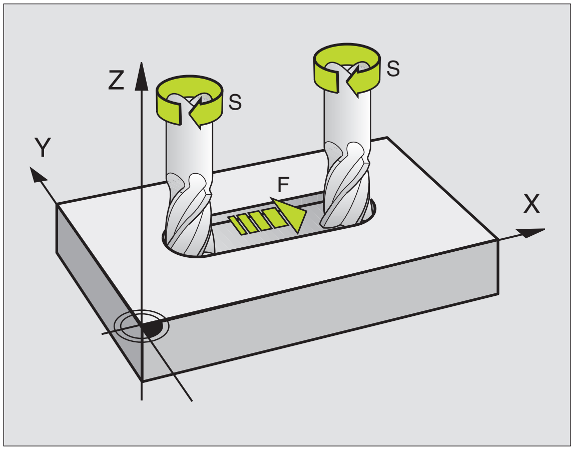


5

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 point moves. The maximum feed rates can be different for the individual axes and are set in machine parameters.

Input

You can enter the feed rate in the **T** block and in every positioning block. (see "Programming tool movements for workpiece machining" on page 211) In millimeter-programs you enter the feed rate in mm/min, and in inch-programs, for reasons of resolution, in 1/10 inch/min.

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 F.

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

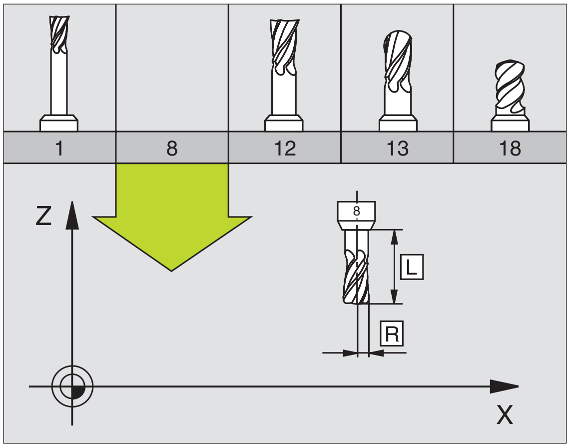
- To program the spindle speed, press the S key on the alphabetic keyboard.
- Enter the new spindle speed.

Changing during program run

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

5.2 Tool Data

Requirements for tool compensation



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

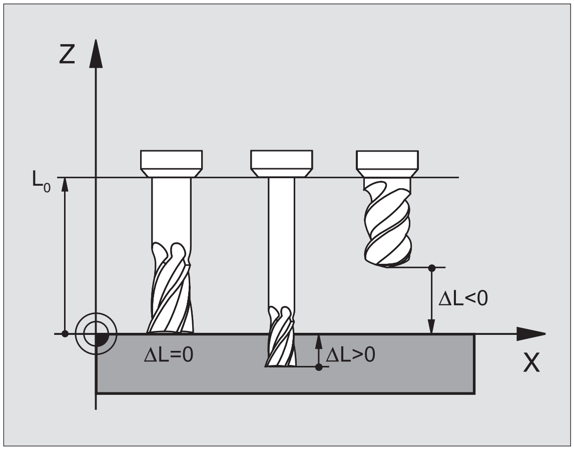
Tool data can be entered either directly in the part program with **G99** or separately in tool tables. In a tool table, you can also enter additional data for the specific tool. The TNC will consider all the data entered for the tool when executing the part program.

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. Tool names can have up to 16 characters.

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

Tool length L



There are two ways to determine the tool length L:

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

For the algebraic sign:

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

To determine the length:

- Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z=0).
- Set the datum in the tool axis to 0 (datum setting).
- Insert the desired tool.
- Move the tool to the same reference position as the zero tool.
- The TNC displays the difference between the current tool and the zero tool.
- Enter the value in the **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

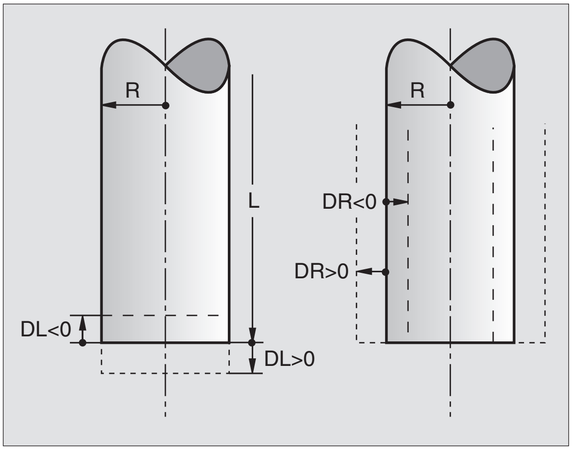
Delta values are offsets in the length and radius of a tool.

A positive delta value describes a tool oversize (**DL**, **DR**, **DR2**>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**, **DR2**<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 **T** 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.

Example

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

Entering tool data in the table

You can define and store up to 30000 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 (Page 188),
- 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 G22 (see "ROUGH-OUT (Cycle G122)" on page 392),
- you want to rough-mill the contour with Cycles G251 to G254 (see "RECTANGULAR POCKET (Cycle G251)" on page 343)
- 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 length L	**Tool length oversize?**
DR	Delta value for tool radius R	**Tool radius oversize?**
DR2	Delta value for tool radius R2	**Tool radius oversize R2?**
LCUTS	Tooth length of the tool for Cycle **G122**	**Tooth length in the tool axis?**
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles **G122**, **G208** and **G251** to **G254**	**Maximum plunge angle?**
TL	Set tool lock (**TL:** for Tool Locked)	**Tool locked?** **Yes = ENT / No = NO ENT**
RT	Number of a replacement tool, if available (**RT:** for Replacement Tool; see also **TIME2**	**Replacement tool?**

Abbr.	Input	Dialog
`TIME1`	Maximum tool life in minutes. This function can vary depending on the machine tool. Your machine manual provides more information.	`Maximum tool age?`
`TIME2`	Maximum tool life in minutes during a T call: If the current tool age exceeds this value, the TNC changes the tool during the next T call of the replacement tool (see also `CUR.TIME`)	`Maximum tool age for TOOL CALL?`
`CUR.TIME`	Current age of the tool in minutes: The TNC automatically counts the current tool life (`CUR.TIME`). A starting value can be entered for used tools.	`Current tool life?`
`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?`
`NMAX`	Limits the spindle speed for this tool. The programmed value is monitored (error message) as well as a shaft speed increase via the potentiometer. Function inactive: Enter **–**	`Maximum speed [rpm]?`
`LIFTOFF`	Definition of whether the TNC should retract the tool in the direction of the positive tool axis at an NC stop in order to avoid leaving dwell marks on the contour. If **Y** is defined, the TNC retracts the tool from the contour by 0.1 mm, provided that this function was activated in the NC program with M148 (see "Automatically retract tool from the contour at an NC stop: M148" on page 268).	`Retract tool Y/N ?`
`P1 ... P3`	Machine-dependent function: Transfer of a value to the PLC. Refer to your machine manual.	`Value?`
`KINEMATIC`	Machine-dependent function: Kinematics description for vertical milling heads, which the TNC adds to the active machine kinematics.	`Additional kinematic description?`
`T-ANGLE`	Point angle of the tool. Is used by the Centering cycle (Cycle G240) in order to calculate the centering depth from the diameter entry.	`Point angle (Type DRILL+CSINK)?`
`PITCH`	Thread pitch of the tool (currently still without function)	`Thread pitch (only type TAP)?`
`AFC`	Control setting for the adaptive feed control AFC that you have defined in the `NAME` column of the AFC.TAB table. Apply the feedback-control strategy with the ASSIGN AFC CONTROL SETTING soft key (3rd soft-key row)	`Feedback-control strategy?`

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 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

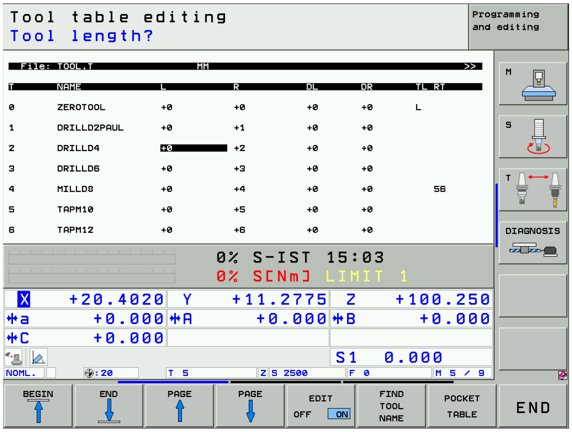
Abbr.	Input	Dialog
`TYPE`	Tool type: Press the ASSIGN TYPE soft key (3rd soft-key row); the TNC superimposes a window where you can select the type of tool. Functions are currently only assigned to the DRILL and MILL tool types.	`Tool type?`
`TMAT`	Tool material: Press the ASSIGN MATERIAL soft key (3rd soft-key row): The TNC superimposes a window where you can select the type of cutting material.	`Tool material?`
`CDT`	Cutting data table: Press the ASSIGN 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. You can only edit TOOL.T 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
Move to beginning of line	BEGIN LINE

Editing functions for tool tables	Soft key
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:

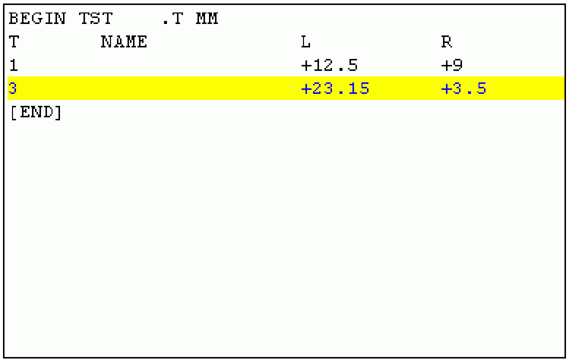
- The target file must exist.
- The file to be copied must contain only the columns (or lines) you want to replace.

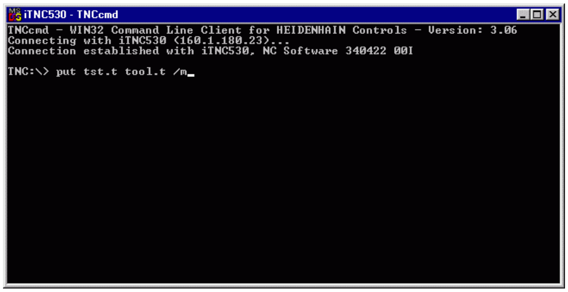
To copy individual columns or lines, press the REPLACE FIELDS soft key (see “Copying a single file” on page 117).

Using an external PC to overwrite individual tool data

The HEIDENHAIN data transfer software TNCremoNT provides an especially convenient way to use an external PC to overwrite tool data (see “Software for data transfer” on page 601). This applies when you measure tool data on an external tool presetter and then want to transfer the data to the TNC. Follow this procedure:

- Copy the tool table TOOL.T to the TNC, for example to TST.T.
- Start the data transfer software TNCremoNT on the PC.
- Establish a connection with the TNC.
- Transfer the copied tool table TST.T to the PC.
- Use any text editor to reduce TST.T to the lines and columns to be changed (see figure). Make sure that the header is not changed and the data is always flush in the column. The tool numbers (column T) need not be consecutive.
- In TNCremoNT, select the menu item <Extras> and <TNCcmd>: This starts TNCcmd.
- To transfer TST.T to the TNC, enter the following command and confirm with the return key (see figure):
  put tst.t tool.t /m





During transfer, only the tool data defined in the subfile (e.g. TST.T) is overwritten. All other tool data of the table TOOL.T remains unchanged.

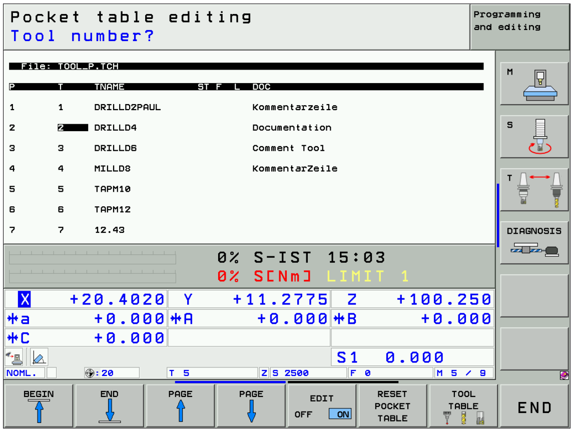
The procedure for copying tool tables using the TNC file manager is described in the section on file management (see “Copying a table” on page 118).

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). In order to be able to manage various magazines in a tool-pocket table (indexing the pocket number), Machine Parameters 7261.0 to 7261.3 must not be equal to 0.

The TNC can control up to **9999 magazine pockets** in the pocket table.



Editing a pocket table in a Program Run operating mode

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

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

- Set the EDIT soft key to ON. On your machine this might not be necessary or even possible. Refer to your machine manual.

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 ST column)	**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	–
PTYP	Tool type. Function is defined by the machine tool builder. The machine tool documentation provides further information.	**Tool type for pocket table?**
P1 ... P5	Function is defined by the machine tool builder. The machine tool documentation provides further information.	**Value?**
RSV	Pocket reservation for box magazines	**Pocket reserv.: Yes = ENT / No = NOENT**
LOCKED_ABOVE	Box magazine: Lock the pocket above	**Lock the pocket above?**
LOCKED_BELOW	Box magazine: Lock the pocket below	**Lock the pocket below?**
LOCKED_LEFT	Box magazine: Lock the pocket at left	**Lock the pocket at left?**
LOCKED_RIGHT	Box magazine: Lock the pocket at right	**Lock the pocket at right?**

Editing functions for pocket tables	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	PAGE
Reset pocket table	RESET POCKET TABLE
Reset tool number column T	RESET COLUMN T
Go to beginning of next line	NEXT LINE
Reset column to original state. Only applies to the columns **RSV**, **LOCKED_ABOVE**, **LOCKED_BELOW**, **LOCKED_LEFT** and **LOCKED_RIGHT**	RESET COLUMN

Calling tool data

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

- Select the tool call function with the TOOL CALL key.

TOOL CALL

- **Tool number:** Enter the number or name of the tool. The tool must already be defined in a **G99** block or in the tool table. The TNC automatically places the tool name in quotation marks. The tool name always refers to the entry in the active tool table TOOL.T. If you wish to call a tool with other compensation values, also enter the index you defined in the tool table after the decimal point.
- **Working spindle axis X/Y/Z:** Enter the tool axis.
- **Spindle speed S:** 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. Instead, you can define the cutting speed Vc in m/min. Press the VC soft 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.
- **Tool length oversize DL:** Enter the delta value for the tool length.
- **Tool radius oversize DR:** Enter the delta value for the tool radius.
- **Tool radius oversize DR2:** Enter the delta value for the tool radius 2.

Example: Tool call

Call tool number 5 in the tool axis Z with a spindle speed of 2500 rpm and a feed rate of 350 mm/min. The tool length is to be programmed with an oversize of 0.2 mm, the tool radius 2 with an oversize of 0.05 mm, and the tool radius with an undersize of 1 mm.

```
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-based (rather than workpiece-based) 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 564).
- Change the tool.
- Resume program run (see "Resuming program run after an interruption," page 567).

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

M101 can vary in function depending on the machine tool. The machine tool manual provides further information.

An automatic tool change with active radius compensation is not possible if an NC program is used on your machine for the tool change. The machine tool manual provides further information.

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

You enter the number of the replacement tool in the **RT** column of the tool table. If no tool number is entered there, the TNC inserts a tool that has the same name as the momentarily active one. The TNC starts the search from the beginning of the tool table and inserts the first tool it finds.

The tool is changed automatically

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

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

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

The TNC does not automatically change the tool as long as a tool change program is running.

Prerequisites for standard NC blocks with radius compensation G40, G41, G42

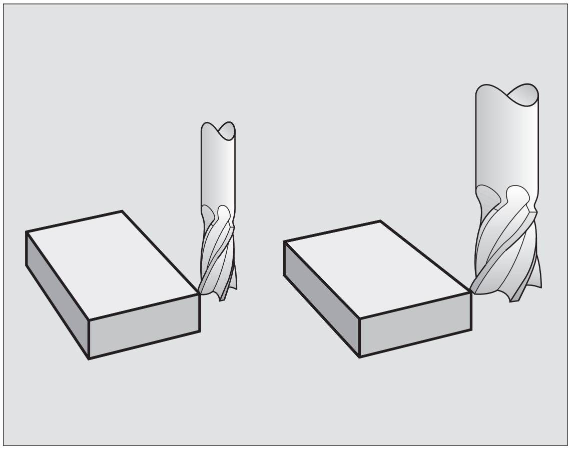
The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool.

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 **TOOL CALL,** the path of the tool in the tool axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

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

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

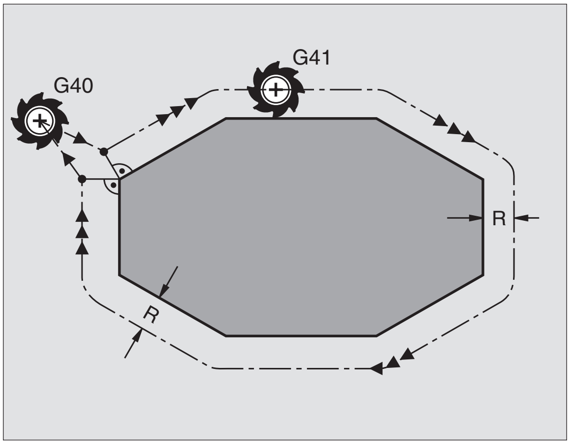
L:	is the tool length **L** from the **G99** block or tool table
$\mathbf{DL}_{\text{TOOL CALL}}$	is the oversize for length **DL** in the **T** block (not taken into account by the position display)
$\mathbf{DL}_{\text{TAB}}$	is the oversize for length **DL** in the tool table.

Tool radius compensation

The NC block for programming a tool movement contains:

- **RL** or **RR** for radius compensation.
- **R+** or **R–** for radius compensation in single-axis movements.
- **R0** if there is no radius compensation.

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



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

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

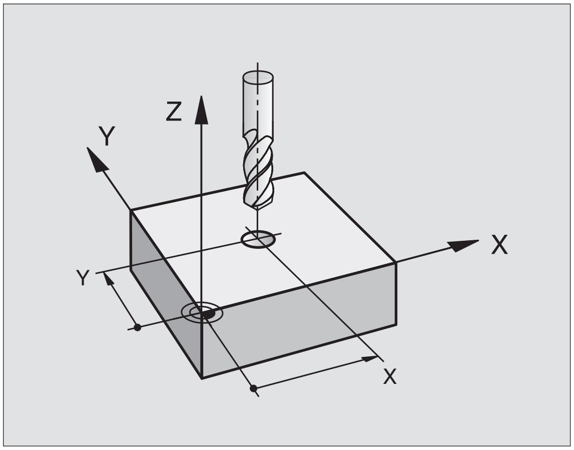
Compensation value = **R** + $\mathbf{DR}_{\text{TOOL CALL}}$ + $\mathbf{DR}_{\text{TAB}}$ where

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

Contouring without radius compensation: R0

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

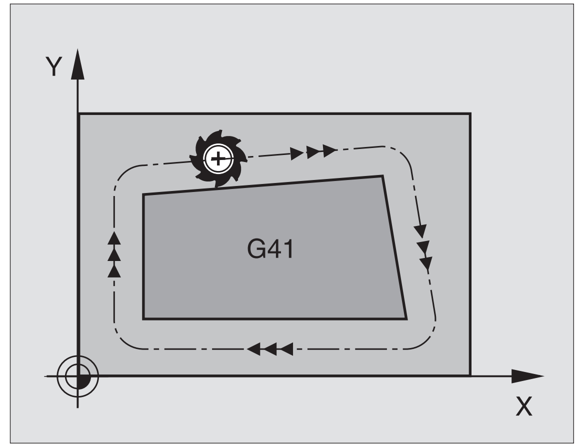
Applications: Drilling and boring, pre-positioning.

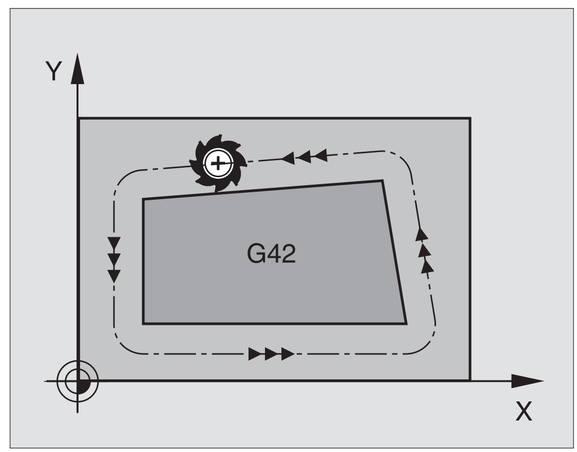


Contouring with radius compensation: G41 and G42

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 too in each following block, since otherwise the TNC will execute the radius compensation in the principal axis again.

Whenever radius compensation is activated with **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

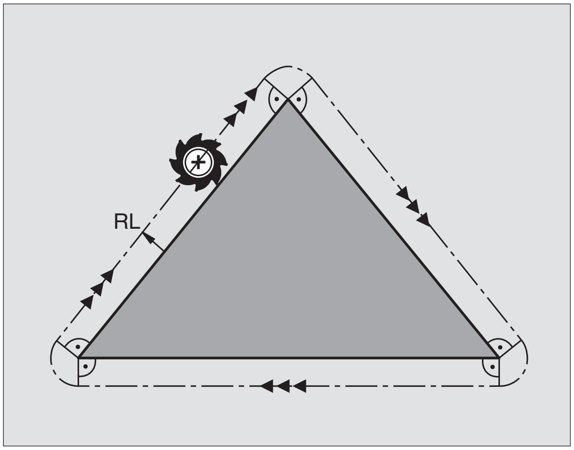
G42 — To select tool movement to the right of the contour, select function G42, or

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

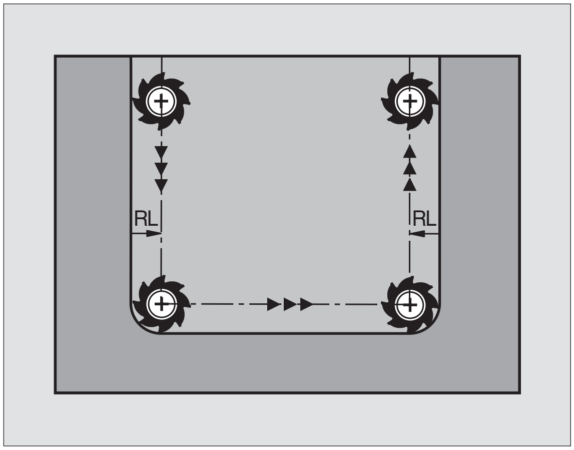
- Outside corners
  If you program radius compensation, the TNC moves the tool around outside corners either on a transitional arc or on a spline (selectable via MP7680). If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.
- 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.

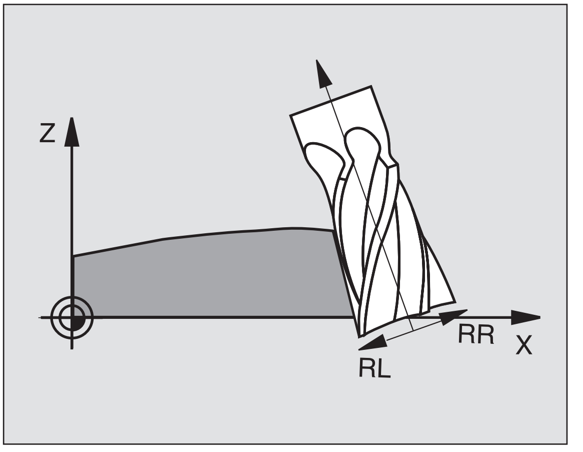
Machining corners without radius compensation

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 255



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 (software option 2)" on page 274) 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.

Note that the TNC makes a compensating movement by the defined **delta values.** The tool radius R defined in the tool table has no effect on the compensation.

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

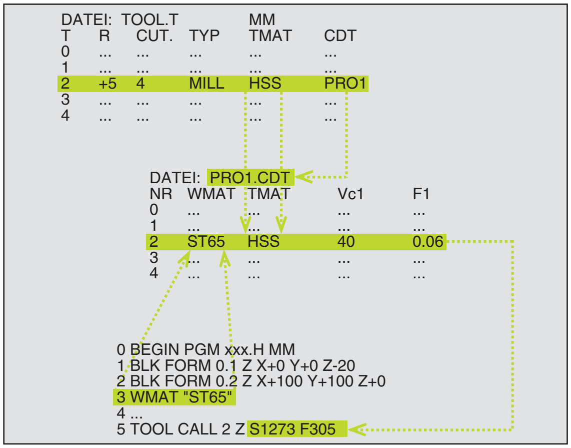
Note

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.

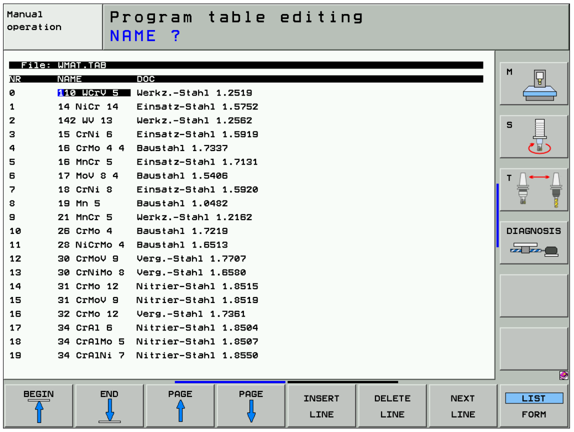


Before you let the TNC automatically calculate the cutting data, the tool table from which the TNC is to take the tool-specific data must be first be activated in the Test Run mode (status S).

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	SORT BlOCK NUMBERS
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). 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 208).
>
> 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:

SPEC FCT

- Show the soft-key row with special functions

WMAT

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

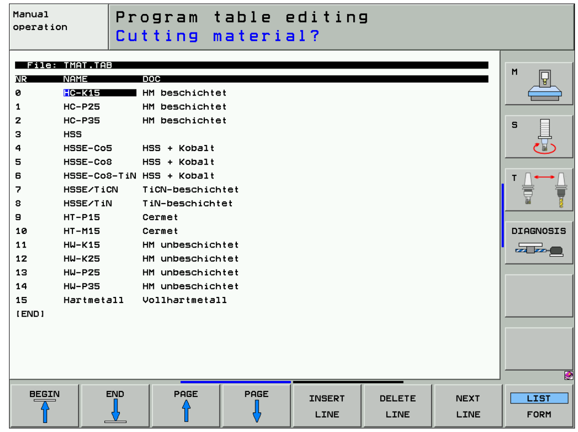
SELECTION WINDOW

- 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). 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.



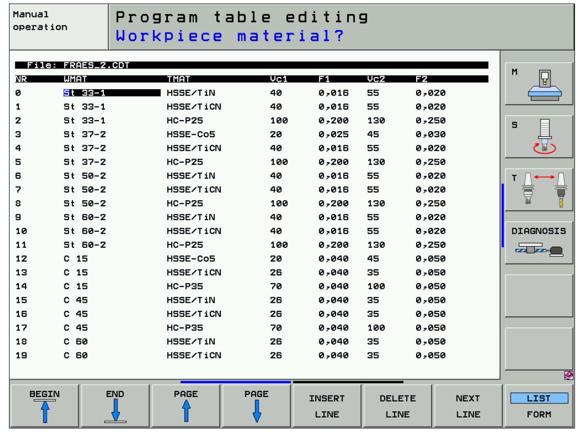
If you change the standard tool cutting material table, you must copy it into a new directory. Otherwise your changes will be overwritten during a software update by the HEIDENHAIN standard data. Define the path in the TNC.SYS file with the code word TMAT= (see “Configuration file TNC.SYS,” page 208).

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. 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 208).

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 opens a standard cutting data table or displays various table formats (machine-dependent). These tables differ from each other in the number of cutting speed/ feed rate combinations they allow. In this case 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

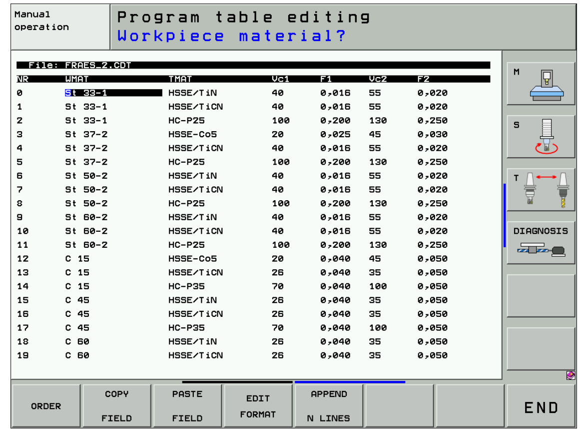
- Tool radius—column R (DR)
- Number of teeth (only with tools for milling)—column CUT
- Tool type—column TYPE
- The tool type influences the calculation of the feed rate:
  Milling tool: $F = S \cdot f_Z \cdot z$
  All other tools: $F = S \cdot f_U$
  S: Spindle speed
  f_Z: Feed per tooth
  f_U: Feed 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 186).

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. You can also switch between table view (default setting) and form view.



> The TNC can process up to 200 characters per row, and up to 30 columns.
>
> If you insert a column into an existing table, the TNC does not automatically shift the values already entered.

Calling the structure editor

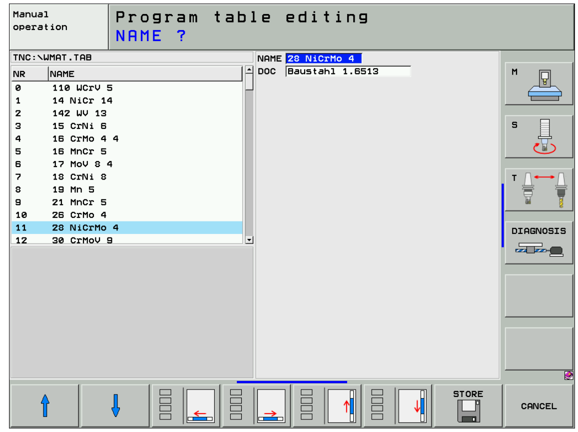
- Press the EDIT FORMAT soft key (2nd soft-key level). The TNC opens the editing window (see figure), in which the table structure is shown rotated by 90°. In other words, a line in the editing window defines a column in the associated table. The meanings of the structure commands (header entries) are 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 decimalplaces.
DEC	Number of decimal places (max. 4, effective only for type N)
ENGLISH to HUNGARIA	Language-dependent dialogs (max. 32 characters)

Switching between table and form view



All tables with the file extension **.TAB** can be opened in either list view or form view.

- Press the FORM LIST soft key. The TNC switches to the view that is not highlighted in the soft key.

In the form view the TNC lists the line numbers with the contents of the first column in the left half of the screen.

In the right half you can change the data.

- Press the key or click in the ENT entry field with the mouse.
- To save any data you have changed, press the END key or the SAVE soft key.
- To discard any changes, press the DEL key or the CANCEL soft key.

The TNC aligns the entry fields on the right side left-justified according to the longest dialog text. If an entry field exceeds the greatest width that can be displayed, a scroll bar appears at the bottom of the window. Use the mouse or soft keys to scroll.

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 206). 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\
```

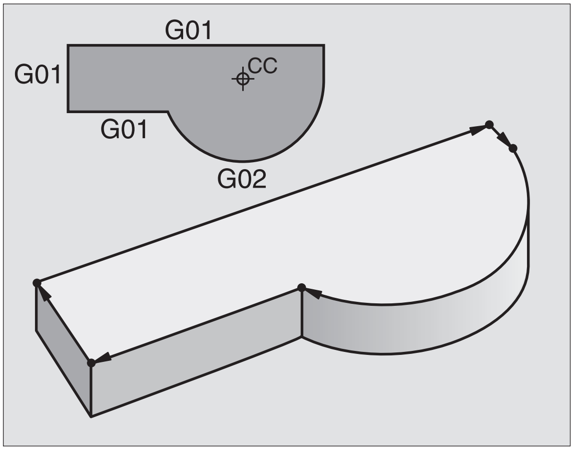
6

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.**

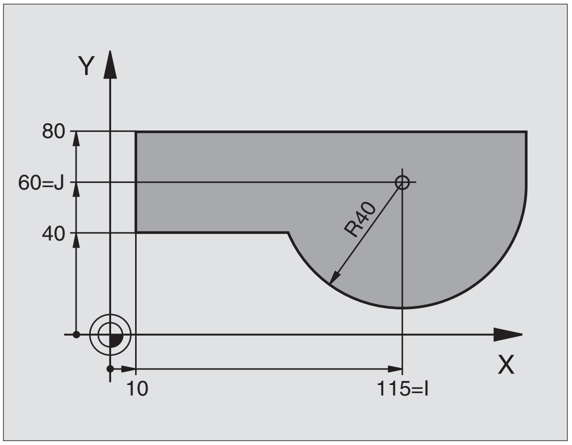


Miscellaneous functions M

With the TNC's miscellaneous functions you can affect:

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- The path 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.

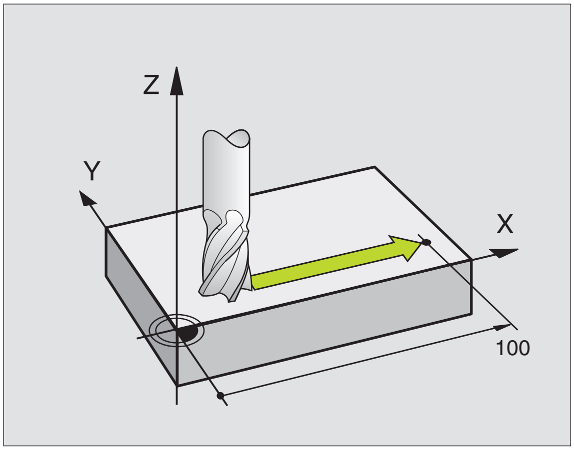
Programming with Q parameters is described in Chapter 10.

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 and from the tool data and radius compensation.

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



Movement parallel to the machine axes

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

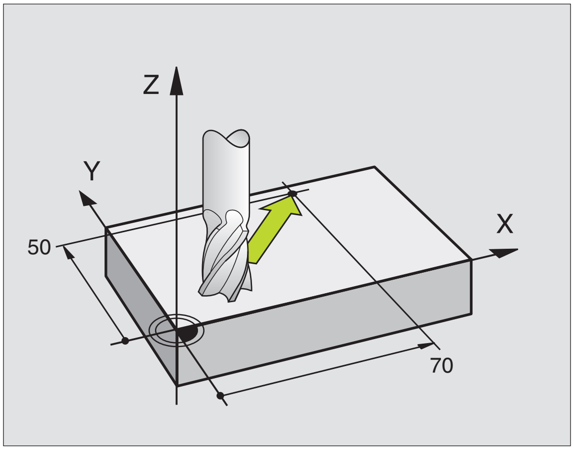
Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

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

N50	Block number
G00	Path function "straight line at rapid traverse"
X+100	Coordinate of the end point

The tool retains the Y and Z coordinates and moves to the position X=100 (see figure at upper right).



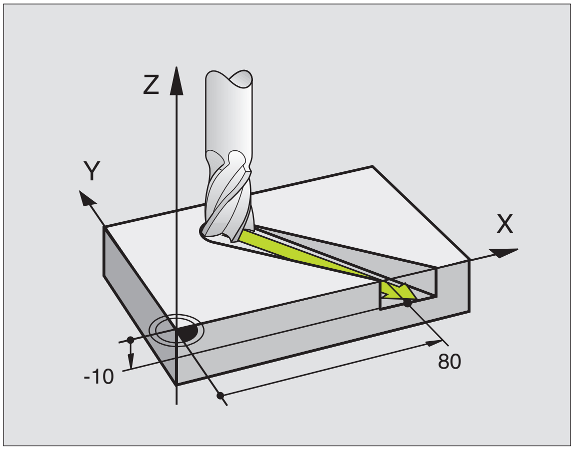
Movement in the main planes

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

Example:

```
N50 G00 X+70 Y+50 *
```

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



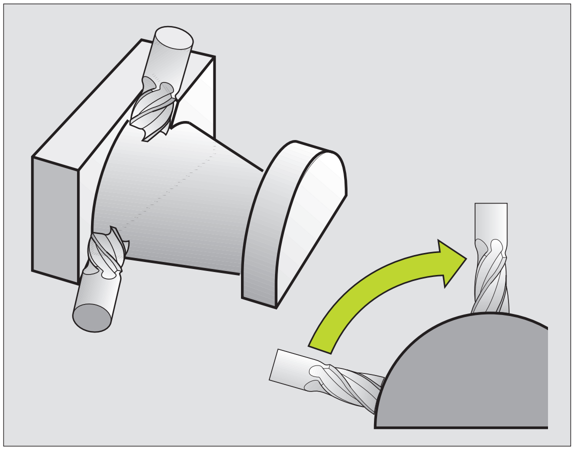
Three-dimensional movement

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

Example:

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

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.

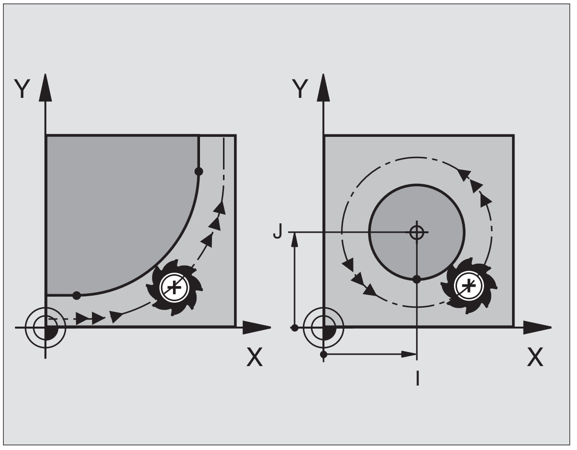
Such programs are too complex to program at the machine, however, and are usually created with a CAD system.

Example:

```
N123 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:

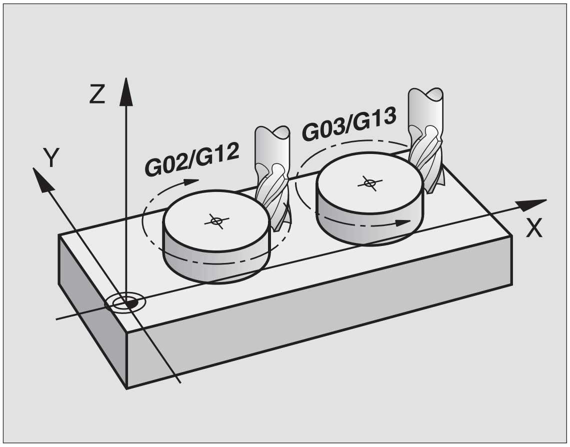
Spindle axis	Main plane	Circle center
Z (G17)	**XY**, also UV, XV, UY	**I, J**
Y (G18)	**ZX**, also WU, ZU, WX	**K, I**
X (G19)	**YZ**, also VW, YW, VZ	**J, K**

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, software option1)," page 448) or Q parameters (see "Principle and Overview," page 506).

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

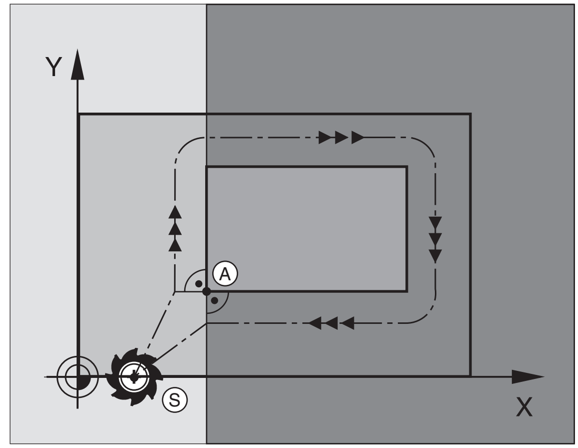
The radius compensation must be in the block in which you move to the first contour element. You cannot begin radius compensation in a circle block. It must be activated beforehand in a straight-line block (see "Path Contours—Cartesian Coordinates," page 218).

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

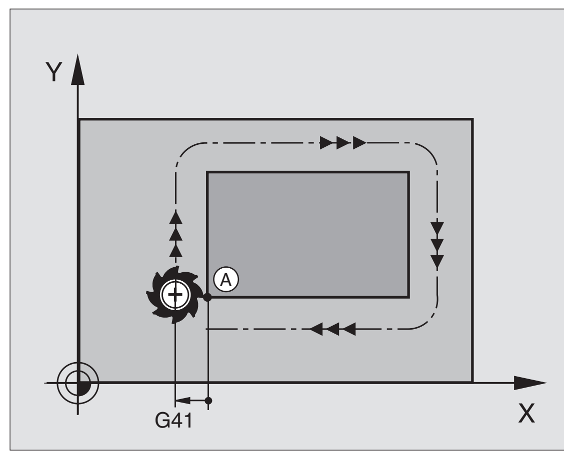


The tool approaches the first contour point from the starting point. The starting point must be:

- 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.

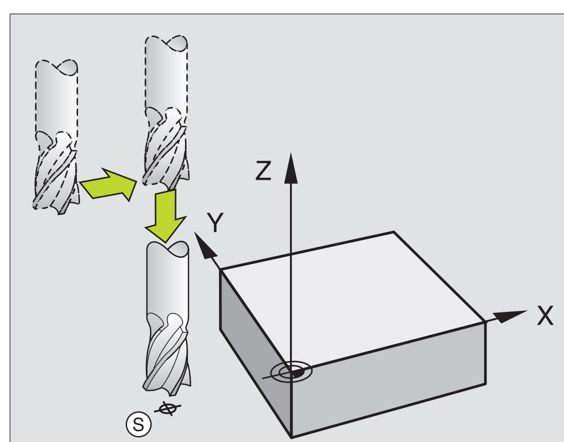


First contour point

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.



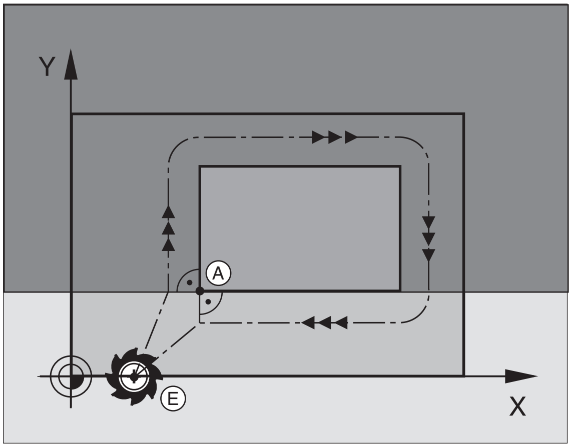
Example NC blocks

```
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.

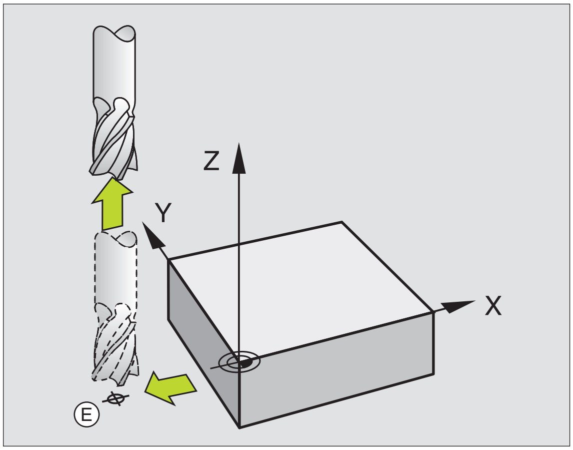


Example

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.



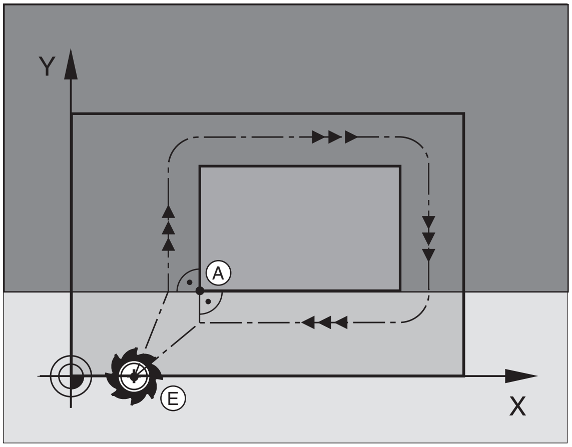
Example NC blocks

```
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.

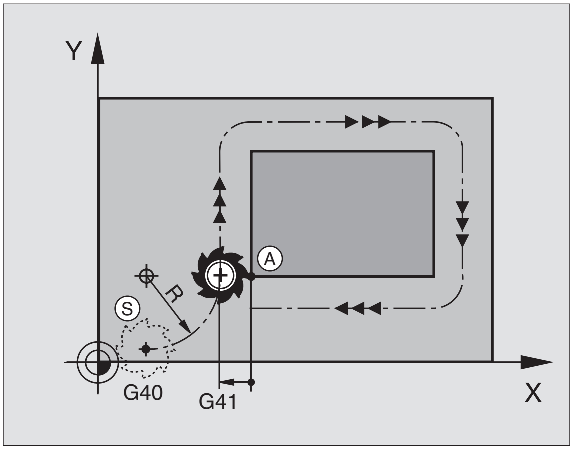


Example

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.

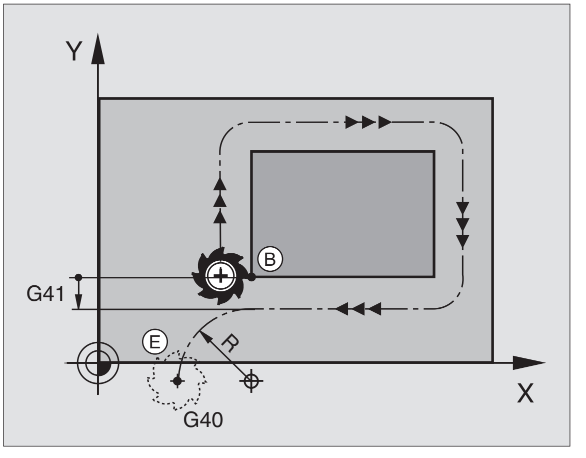
Approach

- **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

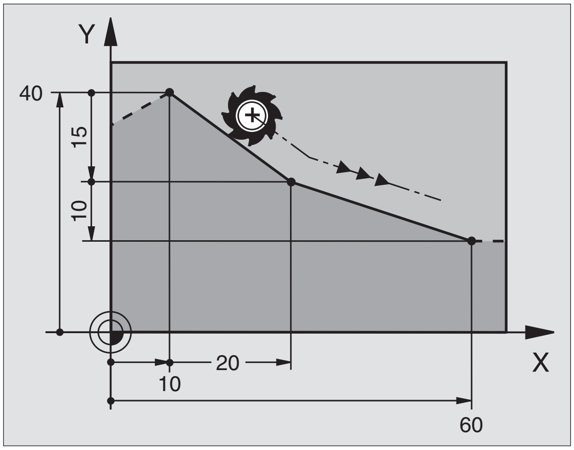
6.4 Path Contours—Cartesian Coordinates

Overview of path functions

Tool movement	Function	Required input	Page
Straight line at feed rate Straight line at rapid traverse	**G00** **G01**	Coordinates of the end points of the straight line	Page 219
Chamfer between two straight lines	**G24**	Length of chamfer **R**	Page 220
–	**I**, **J**, **K**	Coordinates of the circle center	Page 222
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**	Page 223
Circular path corresponding to active direction of rotation	**G05**	Coordinates of the arc end point and circular radius **R**	Page 224
Circular arc with tangential connection to the preceding contour element	**G06**	Coordinates of the arc end point	Page 226
Circular arc with tangential connection to the preceding and subsequent contour elements	**G25**	Rounding radius **R**	Page 221

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 line end point

Further entries, if necessary:

- **Radius compensation G40/G41/G42**
- **Feed rate F**
- **Miscellaneous function M**

Example NC blocks

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

Actual position capture

You can also generate a straight-line block (G01 block) by using the ACTUAL-POSITION-CAPTURE key:

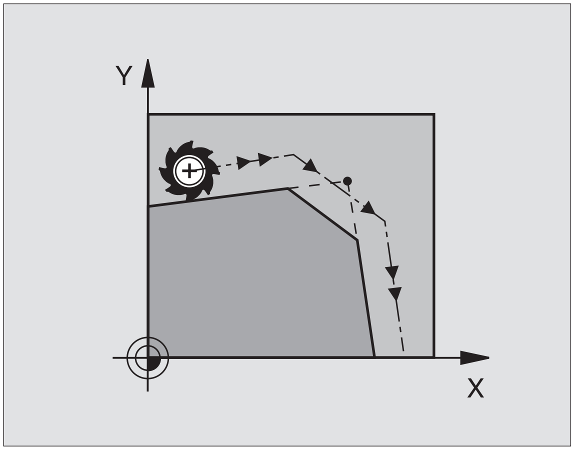
- In the Manual Operation mode, move the tool to the position you wish to capture.
- Switch the screen display to Programming and Editing.
- Select the program block after which you want to insert the block.
- Press the ACTUAL-POSITION-CAPTURE key: The TNC generates an G01 block with the actual position coordinates.

In the MOD function, you define the number of axes that the TNC saves in a G01 block (see “MOD Functions,” page 594).

Inserting a chamfer between two straight lines

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

- The blocks before and after the **G24** block must be in the same working plane.
- The radius compensation before and after the **G24** block must be the same.
- The chamfer must be able to be machined with the current tool.



Programming

G 24

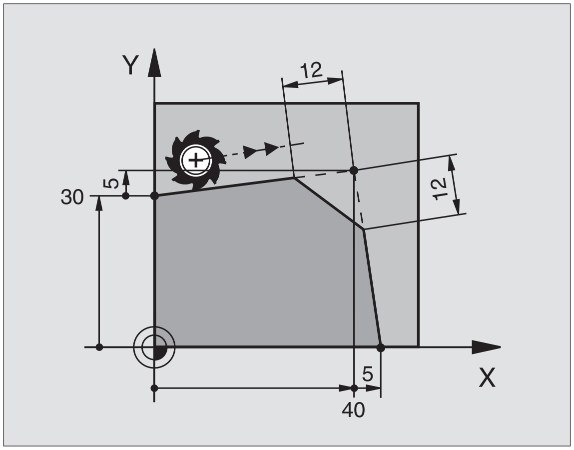
- **Chamfer side length:** Length of the chamfer

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.

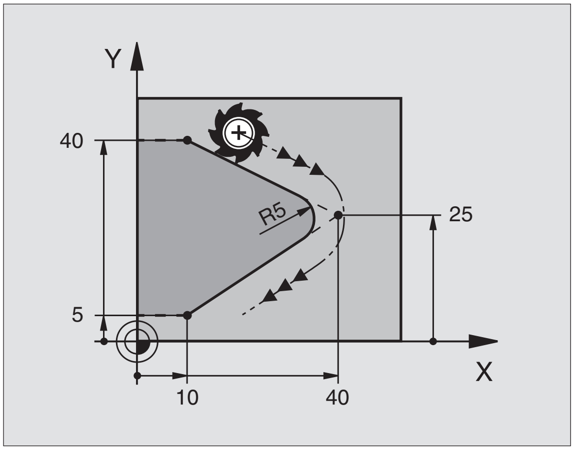
A feed rate programmed in the **G24** block is effective only in that block. After the **G24** block, the previous feed rate becomes effective again.

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 able to be machined with the called tool.



Programming

G 25

▶ **Rounding 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.

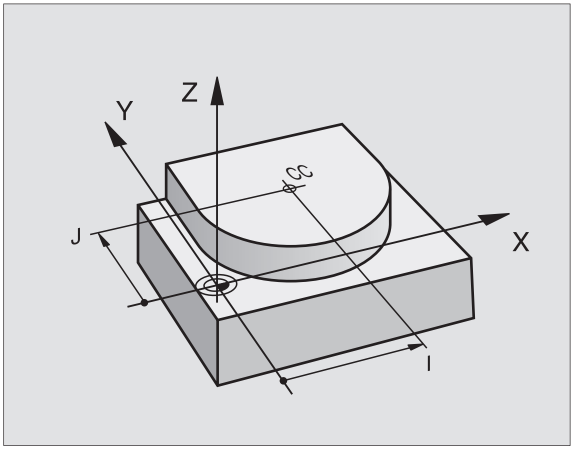
A feed rate programmed in the **G25** block is effective only in that block. After the **G25** block, the previous feed rate becomes effective again.

You can also use a **G25** block for a tangential contour approach (see "Tangential approach and departure," page 216).

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.

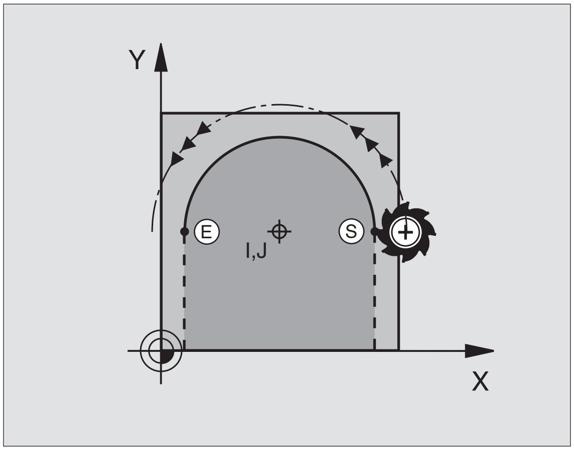
The only effect of **I** and **J** is to define a position as a circle center—the tool does not move to the position.

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

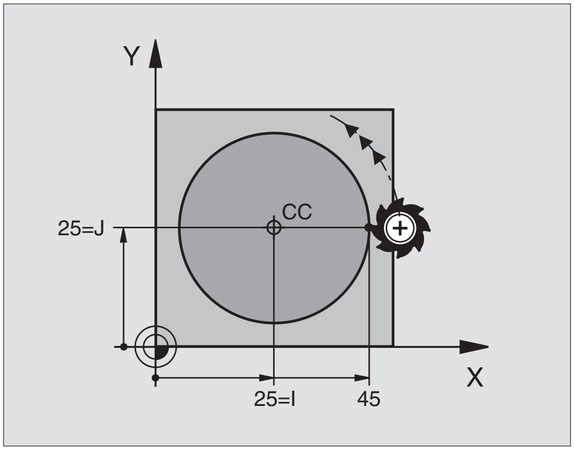
> The TNC normally makes circular movements in the active working plane. If you program circular arcs that do not lie in the active working plane, for example **G2 Z... X...** with a tool axis Z, and at the same time rotate this movement, then the TNC moves the tool in a spatial circular arc, which means a circular arc in 3 axes.

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

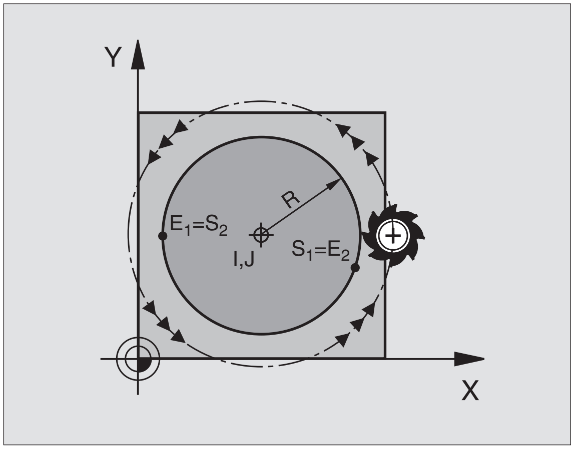
For the end point, enter the same point that you used for the starting point.



> The starting and end points of the arc must lie on the circle.
>
> Input tolerance: up to 0.016 mm (selected with MP7431).
>
> Smallest possible circle that the TNC can traverse: : 0.0016 µm.

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 * (BOGEN 1)
```

or

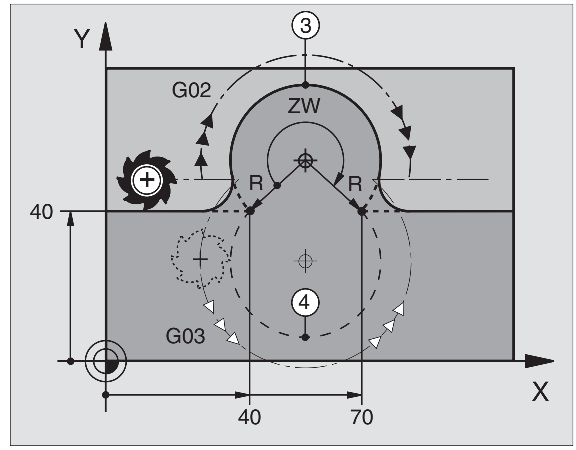
```
N110 G03 X+70 Y+40 R+20 * (BOGEN 2)
```

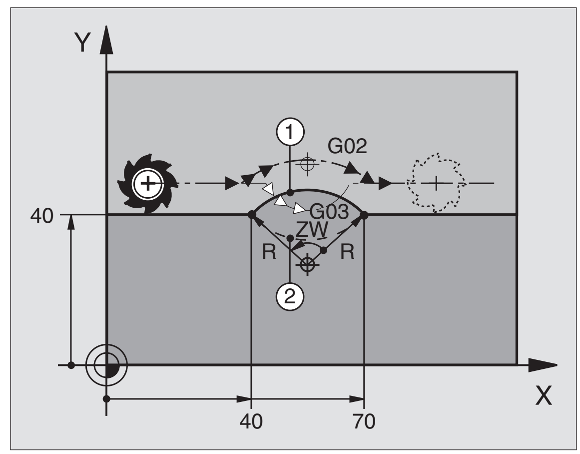
or

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

or

```
N110 G03 X+70 Y+40 R-20 * (BOGEN 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.

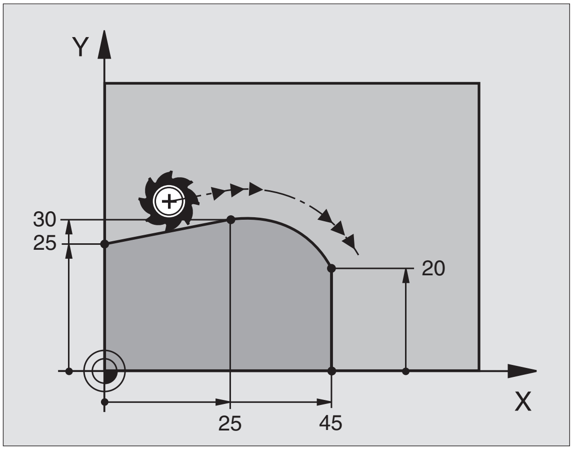
You can also enter rotary axes A, B and C.

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:

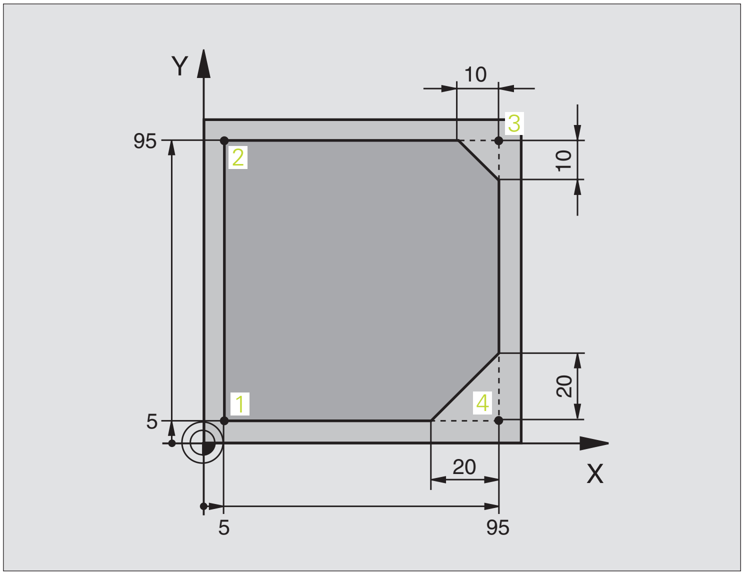
- Feed rate F
- Miscellaneous function M

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 *
```

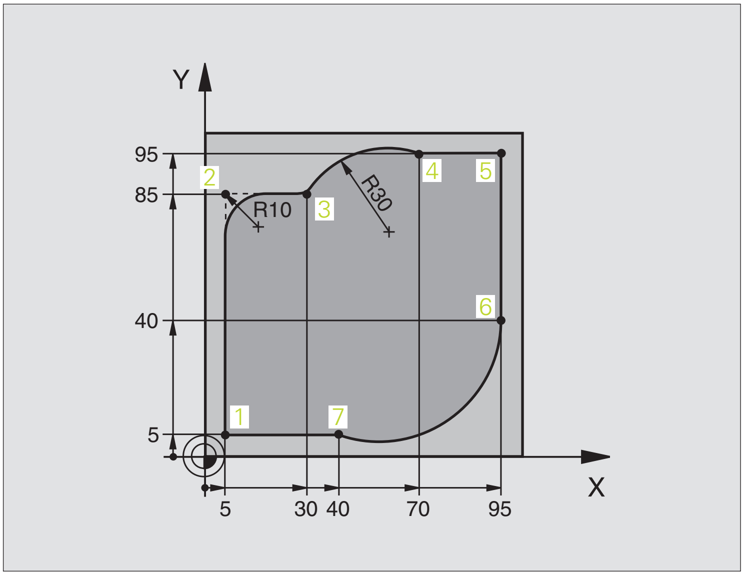
A tangential arc is a two-dimensional operation: the coordinates in the **G06** block and in the contour element preceding it must be in the same plane of the arc.

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
N99999999 %LINEAR G71 *	

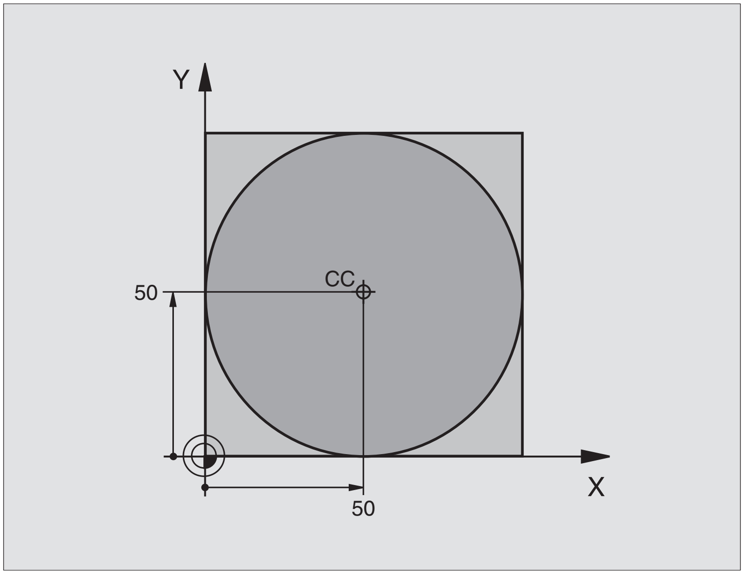
Example: Circular movements with Cartesian coordinates



%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, circular arc 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
N99999999 %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
N99999999 %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 106).

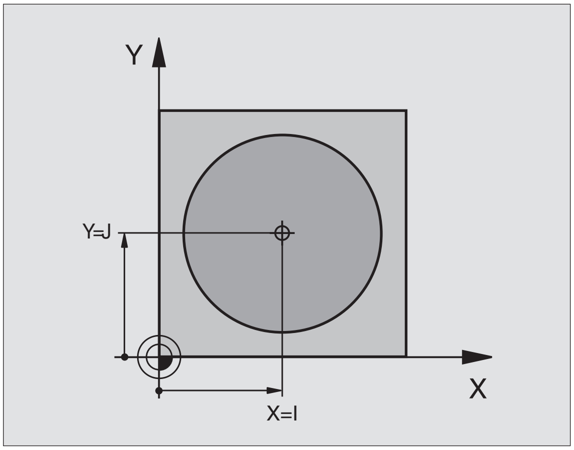
Polar coordinates are useful with:

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

Tool movement	Function	Required input	Page
Straight line at feed rate Straight line at rapid traverse	**G10** **G11**	Polar radius, polar angle of the straight-line end point	Page 232
Circular path in clockwise direction Circular path in counterclockwise direction	**G12** **G13**	Polar angle of the circle end point	Page 232
Circular path corresponding to active direction of rotation	**G15**	Polar angle of the circle end point	Page 232
Circular arc with tangential connection to the preceding contour element	**G16**	Polar radius, polar angle of the arc end point	Page 233

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

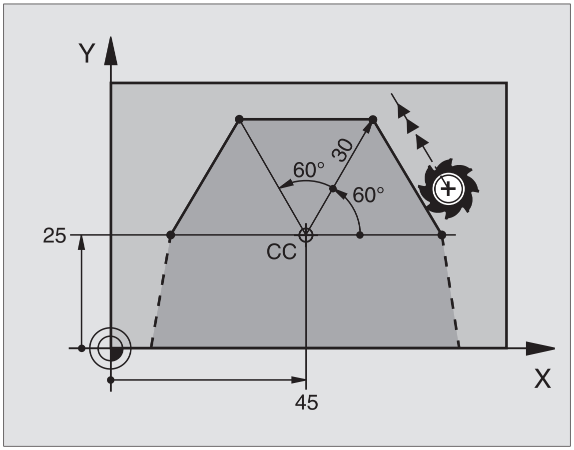
▶ Enter Cartesian coordinates for the pole, or if you want to use the last programmed position, enter **G29.** Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole.

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:

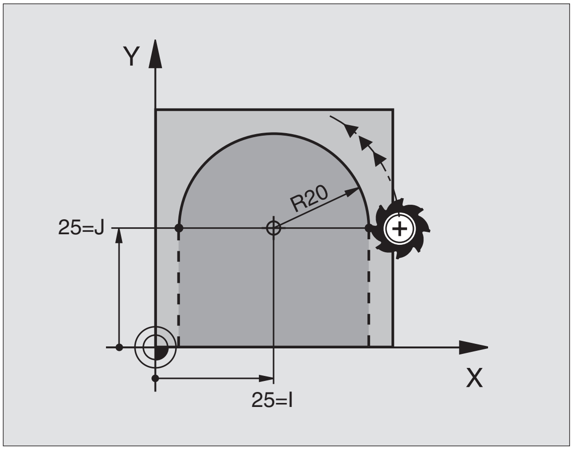
- Angle from angle reference axis to **R** is counterclockwise: **H** >0
- Angle from angle reference axis to **R** is clockwise: **H** <0

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

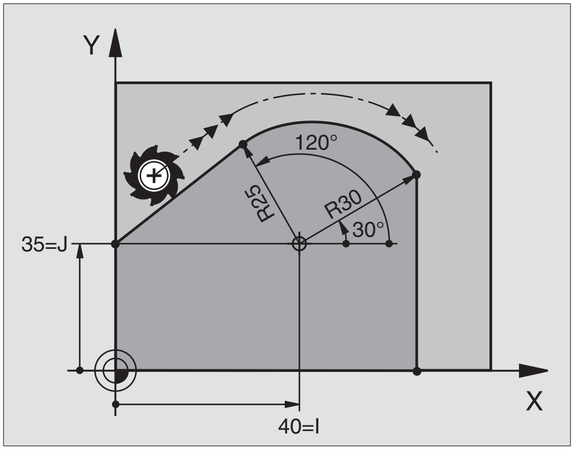
- Polar-coordinates angle **H:** Angular position of the arc end point between –5400° and +5400°

Example NC blocks

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

Circular arc G16 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

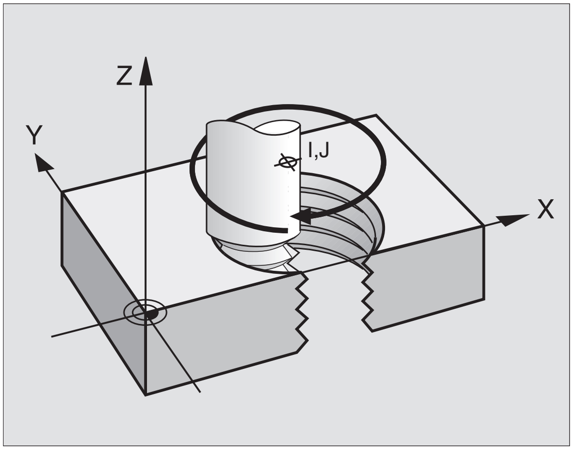
```
N120 I+40 J+35 *
N130 G01 G42 X+0 Y+35 F250 M3 *
N140 G11 R+25 H+120 *
N150 G16 R+30 H+30 *
N160 G01 Y+0 *
```

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

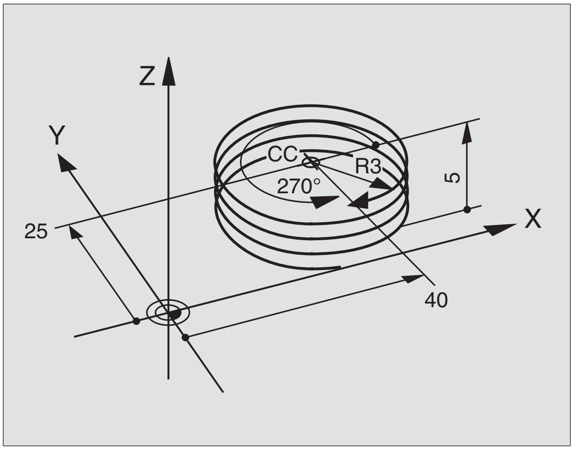
The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

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 492)



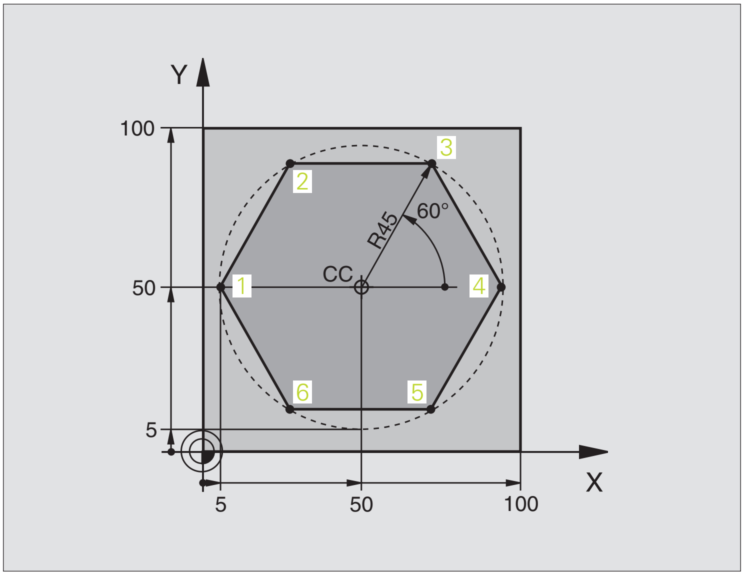
G 12

- 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

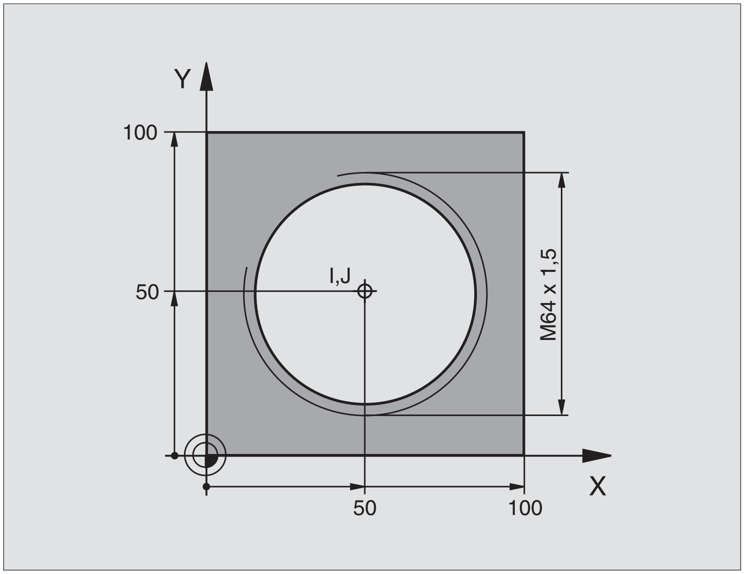
```
N120 I+40 J+25 *
N130 G01 Z+0 F100 M3 *
N140 G11 G41 R+3 H+270 *
N150 G12 G91 H-1800 Z+5 *
```

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
`N100 G26 R5 *`	Approach the contour at point 1
`N110 H+120 *`	Move to point 2
`N120 H+60 *`	Move to point 3
`N130 H+0 *`	Move to point 4
`N140 H-60 *`	Move to point 5
`N150 H-120 *`	Move to point 6
`N160 H+180 *`	Move to point 1
`N170 G27 R5 F500 *`	Tangential departure
`N180 G40 R+60 H+180 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N190 G00 Z+250 M2 *`	Retract in the spindle axis, end of program
`N99999999 %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 *	

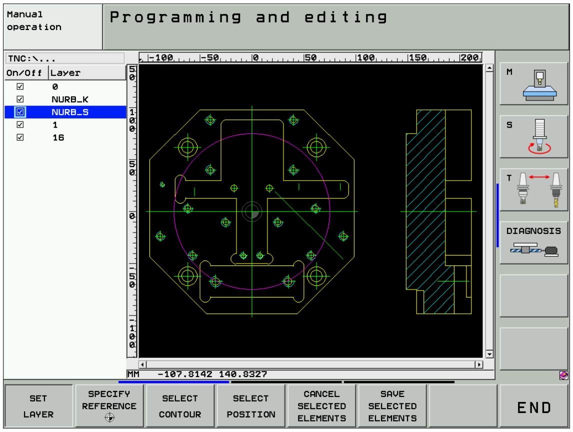
To cut a thread with more than 16 revolutions

...	
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)
N99999999 %HELIX G71 *	

6.6 Generating Contour Programs from DXF Data (Software Option)

Function



DXF files created in a CAD system can be opened directly by the TNC, in order to extract contours or machining positions, and save them as **conversational programs** or as point files. Plain-language programs acquired in this manner can also be run by older TNC controls, since these contour programs only contain **L, CC** and **CP** blocks.

If you process DXF files in the **Programming and Editing** operating mode, the TNC generates contour programs with the **.H** file extension. If you process DXF files in the smarT.NC operating mode, the TNC generates contour programs with the **.HC** file extension.

The DXF files to be processed must be stored on the hard disk of your TNC.

Before loading the file to the TNC, ensure that the name of the DXF file does not contain any blank spaces or illegal special characters.(see “File names” on page 110)

The DXF file to be opened must contain at least one layer.

The TNC supports the most common DXF format, R12 (equivalent to AC1009).

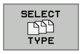
The following DXF elements are selectable as contours:

- LINE (straight line)
- CIRCLE (complete circle)
- ARC (circular arc)

Opening a DXF file

- Select the Programming and Editing operating mode.

- Call the file manager.



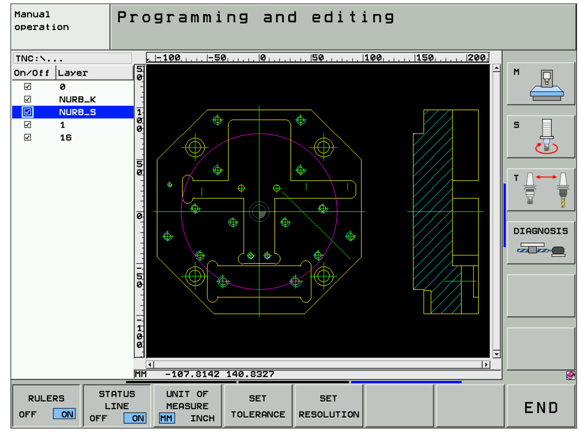
- In order to see the soft-key menu for selecting the file type to be displayed, press the SELECT TYPE soft key.

- In order to show all DXF files, press the SHOW DXF soft key.
- Select the directory in which the DXF file is saved.

- Select the desired DXF file, and load it with the ENT key. The TNC starts the DXF converter and shows the contents of the DXF file on the screen. The TNC shows the layers in the left window, and the drawing in the right window.

Basic settings

The third soft-key row has various possibilities for settings:



Setting	Soft key
Show/hide rulers: The TNC shows the rulers at the left and top edges of the drawing. The values shown on the ruler are based on the drawing datum.	RULERS OFF ON
Show/hide status bar: The TNC shows the status bar at the bottom edge of the drawing. The following information is shown in the status bar: ■ Active unit of measurement (MM or INCH) ■ X and Y coordinates of the current mouse position ■ In the SELECT CONTOUR mode, the TNC shows whether the selected contour is open **`(open contour)`** or closed **`(closed contour)`.**	STATUS LINE OFF ON
Unit of Measure MM/INCH: Enter the unit of measurement of the DXF file. The TNC then outputs the contour program in this unit of measurement.	UNIT OF MEASURE MM INCH
Set Tolerance: The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred when the drawing was made. The default setting depends on the extent of the entire DXF file.	SET TOLERANCE
Set Resolution: The resolution specifies how many decimal places the TNC should use when generating the contour program. Default setting: 4 decimal places (equivalent to resolution of 0.1 µm when the unit of measure MM is active)	SET RESOLUTION

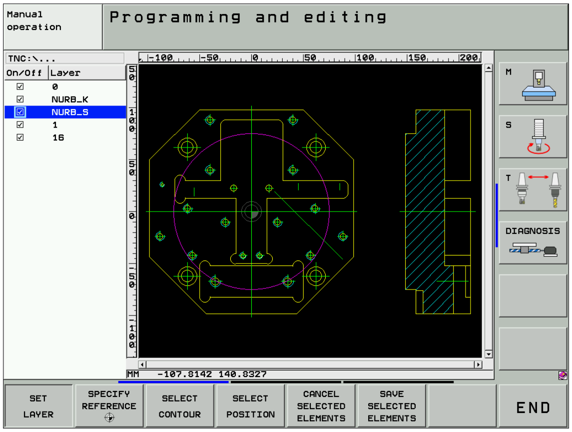
Please note that you must set the correct unit of measurement, since the DXF file does not contain any such information.

If you want to generate programs for older TNC controls, you must limit the resolution to three decimal places. In addition, you must remove the comments that the DXF converter inserts into the contour program.

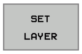
Layer settings

As a rule, DXF files contain multiple layers, with which the designer organizes the drawing. The designer uses the layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

So that as little unnecessary information as possible appears on the screen during selection of the contours, you can hide all excessive layers contained in the DXF file.



> The DXF file to be processed must contain at least one layer.
>
> You can even select a contour if the designer has saved it on different layers.



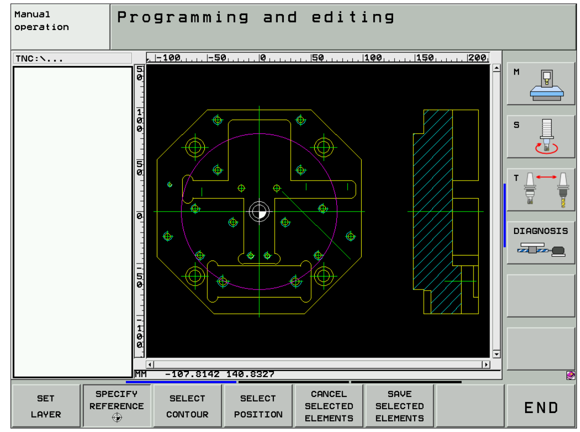
- If it has not already been activated, select the mode for the layer settings. In the left window the TNC shows all layers contained in the active DXF file.
- To hide a layer, select the layer with the left mouse button, and click its checkbox to hide it.
- To show a layer, select the layer with the left mouse button, and click its checkbox again to show it.

Specifying the reference point

The datum of the drawing for the DXF file is not always located in a manner that lets you use it directly as preset for the workpiece. Therefore, the TNC has a function with which you can shift the drawing datum to a suitable location by clicking an element.

You can define a reference point at the following locations:

- At the beginning, end or center of a straight line
- At the beginning or end of a circular arc
- At the transition between quadrants or at the center of a complete circle
- At the intersection of a
  - straight line and a straight line, even if the intersection is actually on the extension of one of the lines
  - straight line and a circular arc
  - straight line and a complete circle
  - circle and circle (regardless of whether a circular arc or a full circle)



> You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to specify a reference point.
>
> You can also change the reference point once you have already selected the contour. The TNC does not calculate the actual contour data until you save the selected contour in a contour program.

Selecting a reference point on a single element

- Select the mode for specifying the reference point.
- Click the element on which you want to set the reference point with the left mouse button. The TNC indicates possible locations for reference points on the selected element with stars.
- Click the star you want to select as reference point. The TNC sets the reference-point symbol to the selected location. Use the zoom function if the selected element is too small.

Selecting a reference point on the intersection of two elements

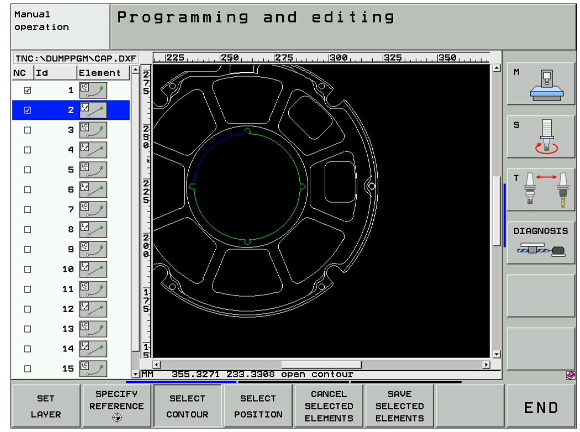
- Select the mode for specifying the reference point.
- Click the first element (straight line, complete circle or circular arc) with the left mouse button. The TNC indicates possible locations for reference points on the selected element with stars.
- Click the second element (straight line, complete circle or circular arc) with the left mouse button. The TNC sets the reference-point symbol on the intersection.

The TNC calculates the intersection of two elements even it is on the extension of one of these elements.

If the TNC calculates multiple intersections, it selects the intersection nearest the mouse-click on the second element.

If the TNC cannot calculate an intersection, it rescinds the marking of the first element.

Selecting and saving a contour



> You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to select a contour.
>
> If you are not using the contour program in the **smarT.NC** operating mode, you must specify the machining sequence when selecting the contour that it matches the desired machining direction.
>
> Select the first contour element such that approach without collision is possible.
>
> If the contour elements are very close to one another, use the zoom function.

SELECT CONTOUR

- Select the mode for choosing a contour. The TNC hides the layers shown in the left window, and the right window becomes active for contour selection.
- To select a contour element, click the desired contour element with the left mouse button. The selected contour element turns blue. At the same time, the TNC marks the selected element with a symbol (circle or line) in the left window.
- To select the next contour element, click the desired contour element with the left mouse button. The selected contour element turns blue. If further contour elements in the selected machining sequence are clearly selectable, these elements turn green. Click on the last green element to assume all elements into the contour program. The TNC shows all selected contour elements in the left window. The TNC displays elements that are still green in the **NC** column without a check mark. This elements are not output to the contour program when it is saved.
- If necessary you can also deselect elements that you already selected, by clicking the element in the right window again, but this time while pressing the CTRL key.

- To save the selected contour elements in a plain-language program, enter any file name in the pop-up window displayed by the TNC. Default setting: Name of the DXF file. If the name of the DXF contains special characters or spaces, the TNC replace the character with an underline.

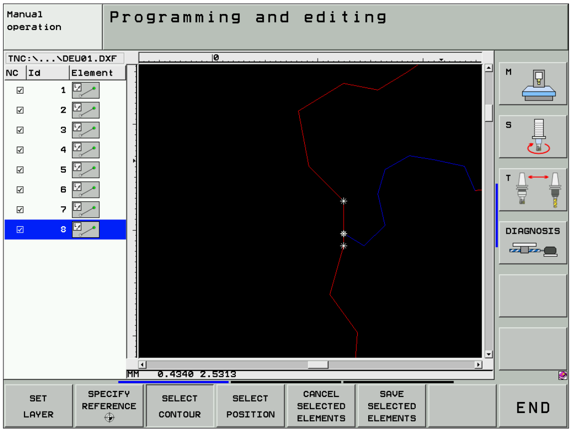
- Confirm the entry: The TNC saves the contour program in the directory in which the DXF file is also saved.

- If you want to select more contours, press the CANCEL SELECTED ELEMENTS soft key and select the next contour as described above.

> The TNC also transfers the workpiece-blank definition **(BLK FORM)** and to the contour program.
>
> The TNC only saves elements that have actually been selected (blue elements), which means that they have been given a check mark in the left window.

Dividing, extending and shortening contour elements

If contour elements to be selected in the drawing do not connect at common end points, then you must first divide the contour element. This function is automatically available if you are in the mode for selecting a contour.

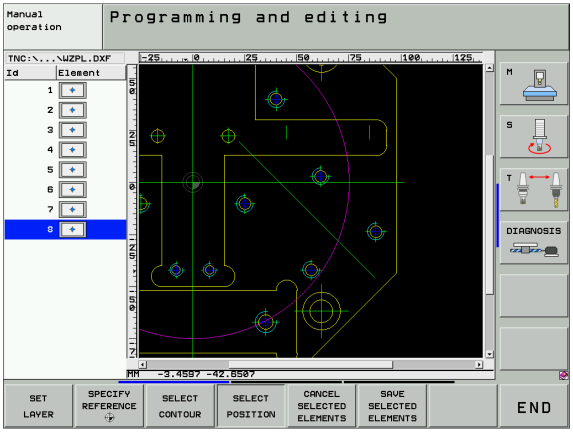


Proceed as follows:

- The poorly connecting contour element is selected, so it is colored blue.
- Click the contour element to be divided: The TNC shows the point of intersection with a star in a circle, and the selectable end points with simple stars.
- Press the CTRL key and click the point of intersection: The TNC divides the contour element at the point of intersection and the stars disappear. If there is a gap, or the elements overlap, the TNC extends or shortens these poorly connecting contour element to the point of intersection of the two elements
- Click the divided contour element again: The TNC shows the end points and points of intersection again.
- Click the desired end point: The TNC now colors the divided element blue.
- Select the next contour element.

> If the contour element to be extended or shortened is a straight line, then the TNC extends the contour element along the same line. If the contour element to be extended or shortened is a circular arc, then the TNC extends the contour element along the same arc.
>
> In order to use this function, at least two contour elements must already be selected, so that the direction is clearly determined.

Selecting and storing machining positions



> You must use the touchpad on the TNC keyboard or a mouse attached via the USB port in order to select a machining position.
>
> If the positions to be selected are very close to one another, use the zoom function.

SELECT POSITION

- Select the mode for choosing a machining position. The TNC hides the layers shown in the left window, and the right window becomes active for position selection.
- In order to select a machining position, click the desired element with the left mouse button. The TNC indicates possible locations for machining positions on the selected element with stars. Click one of the stars: The TNC loads the selected position into the left window (displays a point symbol).
- If you want to specify the machining position at the intersection of two elements, click the first element with the right mouse button: The TNC displays stars at the selectable machining positions.
- Click the second element (straight line, complete circle or circular arc) with the left mouse button. The TNC loads the intersection of the elements into the left window (displays a point symbol).

SAVE SELECTED ELEMENTS

- To save the selected machining positions in a points file, enter any file name in the pop-up window displayed by the TNC. Default setting: Name of the DXF file. If the name of the DXF contains special characters or spaces, the TNC replace the character with an underline.

ENT

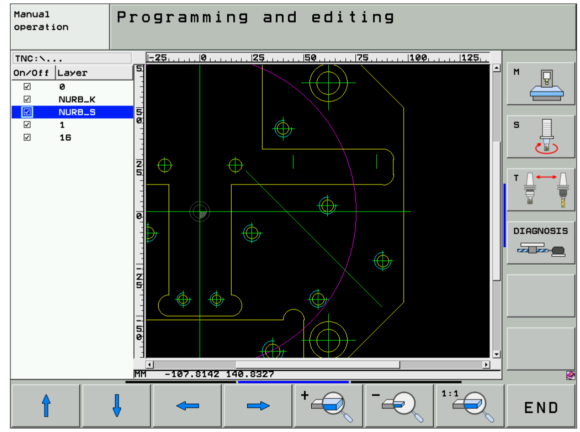
- Confirm the entry: The TNC saves the contour program in the directory in which the DXF file is also saved.

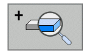
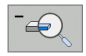
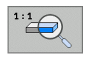
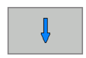
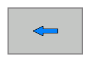
CANCEL SELECTED ELEMENTS

- If you want to select more machining positions in order to save them in a different file, press the CANCEL SELECTED ELEMENTS soft key and select as described above.

Zoom function

The TNC features a powerful zoom function for easy recognition of small details during contour or point selection.



Function	Soft key
Magnify workpiece. The TNC always magnifies the center of the view currently being displayed. Use the scroll bars to position the drawing in the window so that the desired section appears after the soft key has been pressed.	
Reduce workpiece	
Show workpiece at original size	
Move zoomed area upward	
Move zoomed area downward	
Move zoomed area to the left	
Move zoomed area to the right	

If you have a wheel mouse, you can use it to zoom in and out. The zooming center is the location of the mouse pointer.

7

Programming: Miscellaneous Functions

7.1 Entering Miscellaneous Functions M and G38

Fundamentals

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

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- The path 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 or in a separate block. The TNC displays the following dialog question: **Miscellaneous function M ?**

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

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

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

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

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

Entering an M function in a STOP block

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

STOP

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

Example NC blocks

```
87 G38 M6
```

7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

Overview

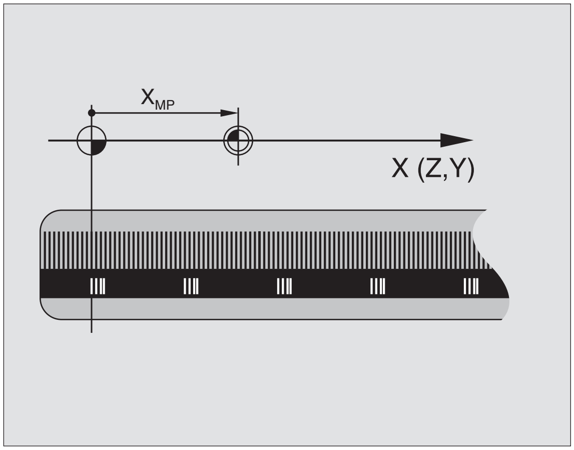
M	Effect	Effective at block... Start	End
M00	Stop program run Spindle STOP Coolant OFF		■
M01	Optional program STOP		■
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (depends on MP7300)		■
M03	Spindle ON clockwise	■	
M04	Spindle ON counterclockwise	■	
M05	Spindle 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 78).

Behavior with M91—Machine datum

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

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

The coordinate values on the TNC screen are shown with respect to the machine datum. Switch the display of coordinates in the status display to REF (see “Status Displays,” page 51).

Behavior with M92—Additional machine datum

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

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

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

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

Effect

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

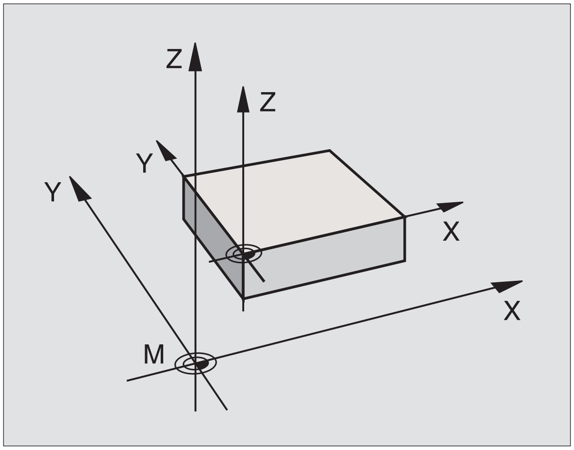
M91 and M92 take effect at the start of block.

Workpiece datum

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

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

The figure 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 614).

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 blockwise in straight-line blocks without tool radius compensation.

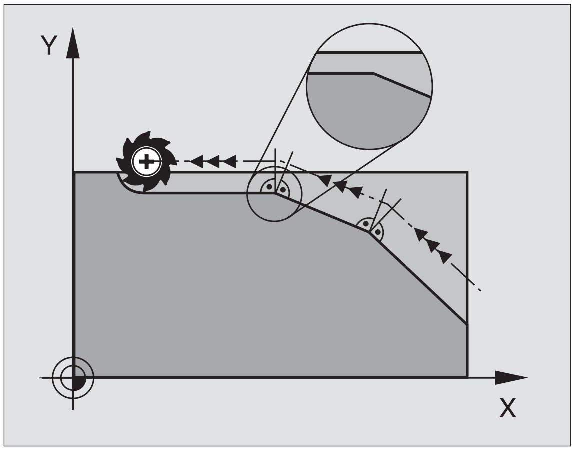
7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

Standard behavior

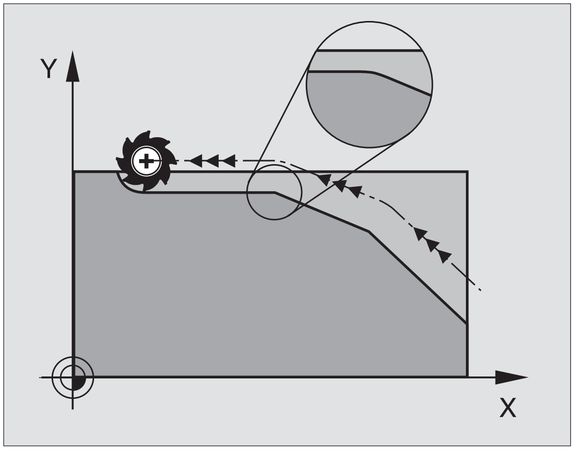
The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an exact stop.

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



Behavior with M90

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



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. However, to define the tolerance for fast contour milling, HEIDENHAIN recommends the use of the TOLERANCE cycle (see “Special Cycles,” page 456).

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 **T** 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 **T.**

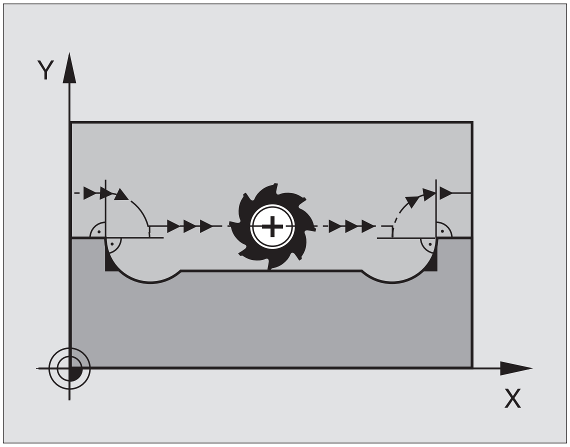
You can also define **T** through Q parameters (see “Principle and Overview” on page 506).

Machining small contour steps: M97

Standard behavior

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

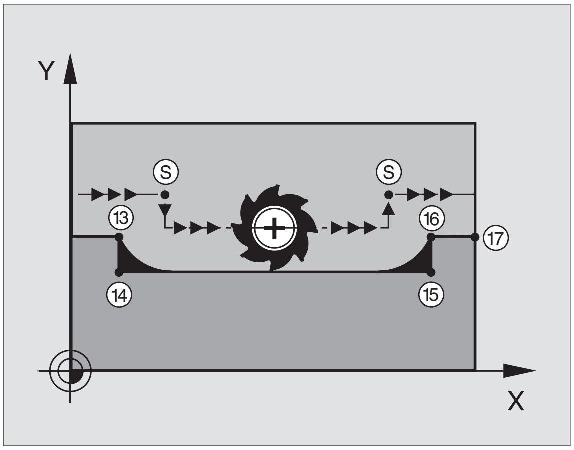
In such cases the TNC interrupts program run and generates the error message “Tool radius too large.”



Behavior with M97

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

Program M97 in the same block as the outside corner.



> Instead of **`M97`** you should use the much more powerful function **`M120 LA`** (see “Calculating the radius-compensated path in advance (LOOK AHEAD): M120” on page 262)!

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

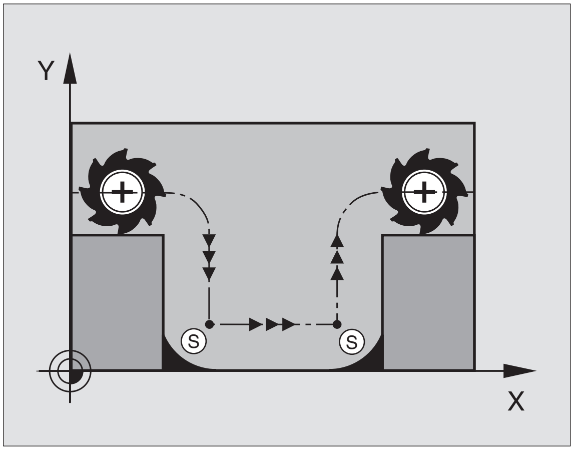
`N50 G99 G01 ... R+20 *`	Large tool radius
`...`	
`N130 X ... Y ... F ... M97 *`	Move to contour point 13
`N140 G91 Y-0.5 ... F ... *`	Machine small contour step 13 to 14
`N150 X+100 ... *`	Move to contour point 15
`N160 Y+0.5 ... F ... M97 *`	Machine small contour step 15 to 16
`N170 G90 X ... Y ...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

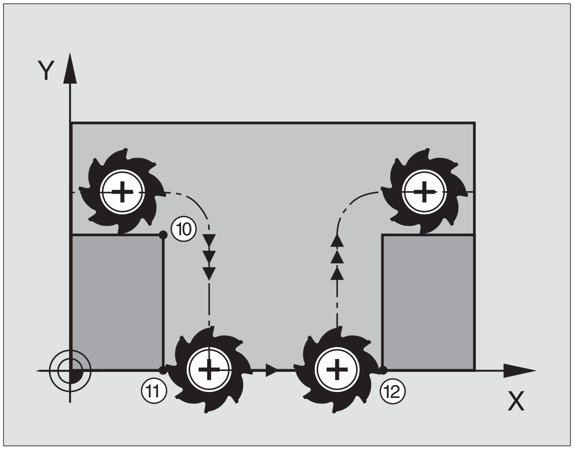
M98 is effective only in the blocks in which it is programmed.

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.

M103 is also effective in an active tilted working plane. The feed rate reduction is then effective during traverse in the negative direction of the **tilted** tool axis.

Example NC blocks

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

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

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 for circular arcs: M109/M110/M111

Standard behavior

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

Behavior at circular arcs with M109

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

Behavior at circular arcs with M110

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

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

Effect

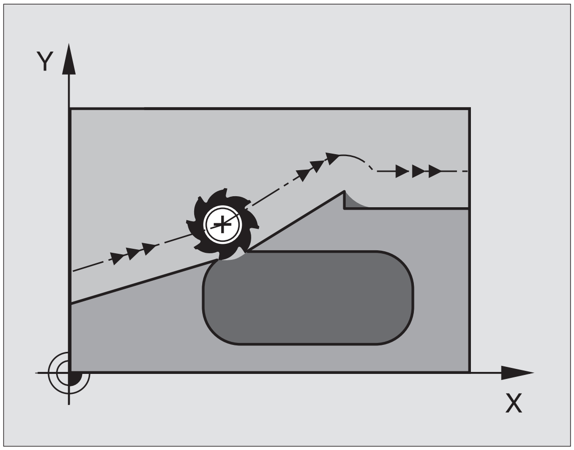
M109 and M110 become effective at the start of block.
To cancel M109 and M110, enter M111.

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

Standard behavior

If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. M97 (see "Machining small contour steps: M97" on page 257) inhibits the error message, but this results in dwell marks and will also move the corner.

If the programmed contour contains undercuts, the tool could 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 (look-ahead) 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 RL or RR. M120 is then effective from this block until

- radius compensation is canceled, or
- M120 LA0 is programmed, or
- M120 is programmed without LA, or
- another program is called with PGM CALL, or
- the working plane is tilted with Cycle G80 or the PLANE function.

M120 becomes effective at the start of block.

Limitations

- After an external or internal stop, you can re-enter the contour with M120 only with the RESTORE POS. AT N function.
- If you are using the path functions G25 and G24, the blocks before and after G25 or G26 must contain only coordinates of the working plane.
- Before using the functions listed below, you have to cancel M120 and the radius compensation:
  - Cycle G60 Tolerance
  - Cycle G80 Working Plane
  - M114
  - M128
  - M138
  - M144
  - PLANE function
  - TCPM FUNCTION (only conversational)
  - WRITE TO KINEMATIC (only conversational format)

Superimposing handwheel positioning during program run: M118

Standard behavior

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

Behavior with M118

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

Input

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

Effect

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

M118 becomes effective at the start of block.

Example NC blocks

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 and in the rotary axis B by ±5° from the programmed value:

```
N250 G01 G41 X+0 Y+38.5 F125 M118 X1 Y1 B5 *
```

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 TRAVERSE 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 M140

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

Input

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

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

Effect

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

M140 becomes effective at the start of the block.

Example NC blocks

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

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

```
N45 G01 X+0 Y+38.5 F125 M140 MB50 *
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 the **FN18: SYSREAD ID230 NR6** function you can find the distance from the current position to the limit of the traverse range in the positive tool axis.

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

When dynamic collision monitoring (DCM) is active, the TNC might move the tool only until it detects a collision and, from there, complete the NC program without any error message. This can result in tool paths different from those programmed!

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).
- Define 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.

The function **M142** is not permitted during a mid-program startup.

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.

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

Effect

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

M143 becomes effective at the start of the block.

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

Standard behavior

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

Behavior with M148

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

The TNC retracts the tool by 0.1 mm in the direction of the tool axis if, in the **LIFTOFF** column of the tool table, you set the parameter **Y** for the active tool (see “Tool table: Standard tool data” on page 183).

> Remember that, especially on curved surfaces, the surface can be damaged during return to the contour. Back the tool off before returning to the contour!

Effect

M148 remains in effect until deactivated with M149.

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

Suppress limit switch message: M150

Standard behavior

The TNC stops program run with an error message if the tool were to leave the active working space during a positioning block. The error message is output before the positioning block is executed.

Behavior with M150

If the end point of a positioning block with M150 is outside the current working space, the TNC moves the tool to the edge of the working space, and then continues the program run without an error message.

Danger of collision!

Keep in mind that the approach path to the position programmed after the M150 block might be changed significantly!

M150 is also effective on traverse range limits defined with the MOD function.

When dynamic collision monitoring (DCM) is active, the TNC might move the tool only until it detects a collision and, from there, complete the NC program without any error message. This can result in tool paths different from those programmed!

Effect

M150 is effective only in the block in which it is programmed.

M150 becomes effective at the start of block.

7.5 Miscellaneous Functions for Rotary Axes

Feed rate in mm/min on rotary axes A, B, C: M116 (software option 1)

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 MPs 7510 and following by the machine tool builder.

M116 works only on rotary tables. M116 cannot be used with swivel heads. If your machine is equipped with a table/head combination, the TNC ignores the swivel-head rotary axes.

M116 is also effective in an active tilted working plane.

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

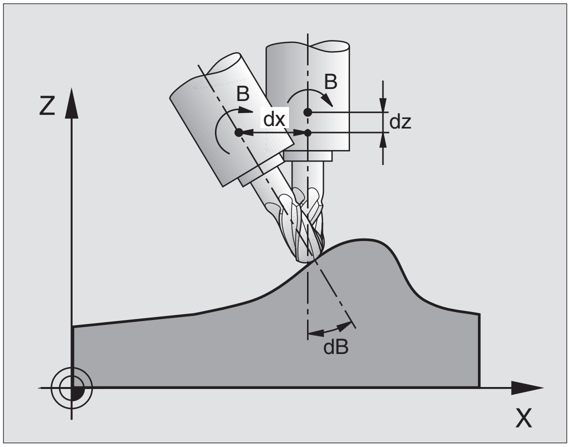
M94 is effective only in the block in which it is programmed.

M94 becomes effective at the start of block.

Automatic compensation of machine geometry when working with tilted axes: M114 (software option 2)

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

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

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

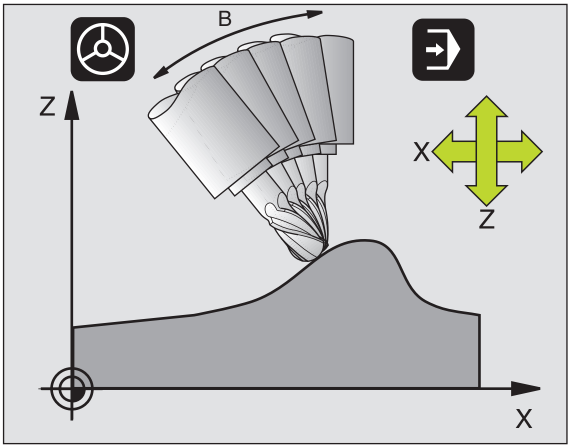
M114 becomes effective at the start of block, M115 at the end of block. M114 is not effective when tool radius compensation is active.

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 (software option 2)

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 (TCPM: Tool Center Point Management)

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

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 **M128** in conjunction with **M118.** Handwheel positioning in a fixed machine 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 when disengaging from the coupling.

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 **TOOL CALL.**

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 the symbol in the status display.

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 geometrical configurations.

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.

```
N50 G01 G41 X+0 Y+38.5 IB-15 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 628).

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:

```
N50 G00 Z+100 R0 M138 C *
```

Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block: M144 (software option 2)

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 defined by the machine tool builder in MPs 7502 and following. The machine tool builder decides upon the behavior of the machine 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 9 999 V

Effect

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

Output voltage as a function of distance: M201

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 9 999 V

Effect

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

Output voltage as a function of speed: M202

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

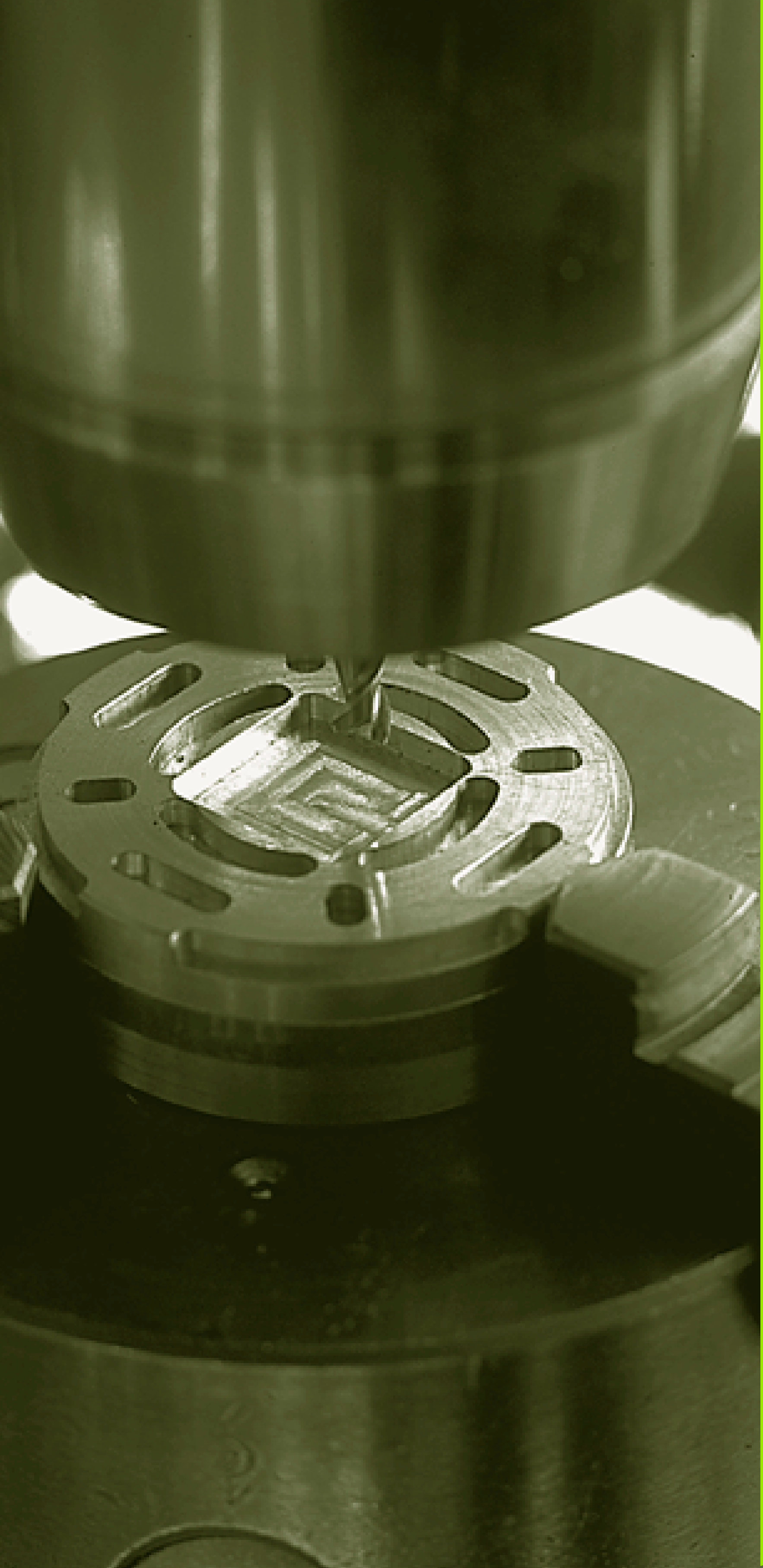
The TNC outputs a programmed voltage as a pulse with a programmed duration TIME.

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 560).

Machine-specific cycles

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

- Cycles G300 to G399
  Machine-specific cycles that are to be defined through the CYCLE DEF key
- Cycles G500 to G599
  Machine-specific cycles that are to be defined through the TOUCH PROBE key

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

Sometimes, machine-specific cycles also use transfer parameters, which HEIDENHAIN already used in the standard cycles. The TNC executes DEF-active cycles as soon as they are defined (See also “Calling a cycle” on page 285) It executes CALL-active cycles only after they have been called (See also “Calling a cycle” on page 285). When DEF-active cycles and CALL-active cycles are used simultaneously, it is important to prevent overwriting of transfer parameters already in use. Use the following procedure:

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

Defining a cycle using soft keys

CYCL DEF

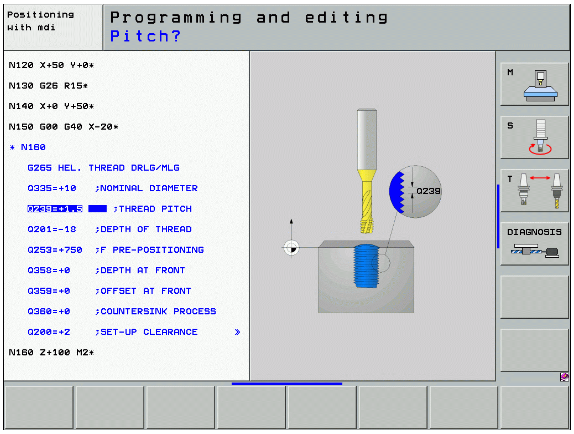
- The soft-key row shows the available groups of cycles.

DRILLING/ THREAD

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

200

- Select a cycle, e.g. DRILLING. The TNC initiates the programming dialog and asks for 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	Page
Cycles for pecking, reaming, boring, counterboring, tapping and thread milling	DRILLING/ THREAD	Page 292
Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS	Page 342
Cycles for producing point patterns, such as circular or linear hole patterns	PATTERN	Page 376
SL (Subcontour List) cycles which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation	SL CYCLES	Page 383
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING	Page 424
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.	Page 438
Special cycles such as dwell time, program call, oriented spindle stop and tolerance	SPECIAL CYCLES	Page 456

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 point patterns on circles and Cycle G221 for point patterns on lines
- 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 288).

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

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

The TNC moves using positioning logic to the position defined in the **CYCL CALL POS** block.

- If the current position in the tool axis is greater than the top surface of the workpiece (Q203), the iTNC moves the tool to the programmed position first in the machining plane and then in the tool axis.
- If the current tool position in the tool axis is below the top surface of the workpiece (Q203), the TNC moves the tool to the programmed position first in the tool axis to the clearance height and then in the working plane to the programmed position.

Three coordinate axes must always be programmed in the **G79:G01** block. With the coordinate in the tool axis you can easily change the starting position. It serves as an additional datum shift.

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 position defined in the cycles serves as an additional shift on the position defined in the **G79:G01** block. You should therefore always define the start position to be set in the cycle as 0.

Calling a cycle with M99/89

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

If the 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.
- In Cycles G77/G78 (CIRCULAR POCKET), G251 (RECTANGULAR POCKET), G252 (CIRCULAR POCKET),G253 (SLOT) and G254 (CIRCULAR SLOT), the TNC machines the cycle in the axis that you programmed in the last positioning block before the cycle call. When tool axis Z is active, the following combinations are permissible:
  - X/Y
  - X/V
  - U/Y
  - U/V

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 INSERT LINE soft key, 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.

Hiding single points from the machining process

In the **FADE** column of the point table you can specify if the defined point is to be hidden during the machining process (see "Optional Block Skip" on page 574).

In the table, select the point to be hidden.

Select the FADE column

ENT Activate the hiding, or

NO ENT Deactivate the hiding

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 END 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, where the tool is located during cycle call. A clearance height or 2nd set-up clearance that is defined separately in a cycle must not be greater than the clearance height defined in the global pattern.

If you want the 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 260).

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 G208 and G262 to G267

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 G210 to G215

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

Effect of the point tables with Cycles G251 to G254

The TNC interprets the points of the working plane as coordinates of the cycle starting point. 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.

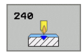
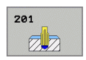
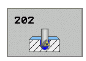
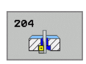
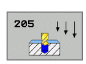
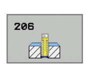
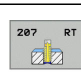
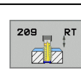
Applies to all 2xx cycles

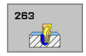
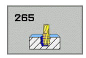
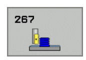
As soon as the current tool axis position lies below the clearance height in with **G79 PAT** the TNC displays the error message **PNT: Clearance plane too small.** The clearance height is calculated from the sum of the workpiece surface coordinate (Q203) and the 2nd set-up clearance (Q204, or set-up clearance Q200, if the value of Q200 is greater than Q204).

8.3 Cycles for Drilling, Tapping and Thread Milling

Overview

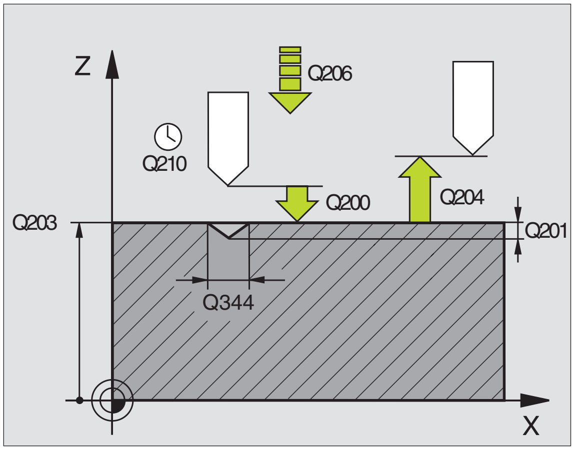
The TNC offers 16 cycles for all types of drilling operations:

Cycle	Soft key	Page
G240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth		Page 294
G200 DRILLING With automatic pre-positioning, 2nd set-up clearance		Page 296
G201 REAMING With automatic pre-positioning, 2nd set-up clearance		Page 298
G202 BORING With automatic pre-positioning, 2nd set-up clearance		Page 300
G203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing		Page 302
G204 BACK BORING With automatic pre-positioning, 2nd set-up clearance		Page 304
G205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance		Page 306
G208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance		Page 309
G206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		Page 311
G207 RIGID TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		Page 313
G209 TAPPING W/ CHIP BRKG Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking		Page 315

Cycle	Soft key	Page
G262 THREAD MILLING Cycle for milling a thread in pre-drilled material		Page 319
G263 THREAD MLLNG/CNTSNKG Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer		Page 321
G264 THREAD DRILLING/MLLNG Cycle for drilling into the solid material with subsequent milling of the thread with a tool		Page 325
G265 HEL.THREAD DRLG/MLG Cycle for milling the thread into the solid material		Page 329
G267 OUTSIDE THREAD MLLNG Cycle for milling an external thread and machining a countersunk chamfer		Page 333

CENTERING (Cycle 240)

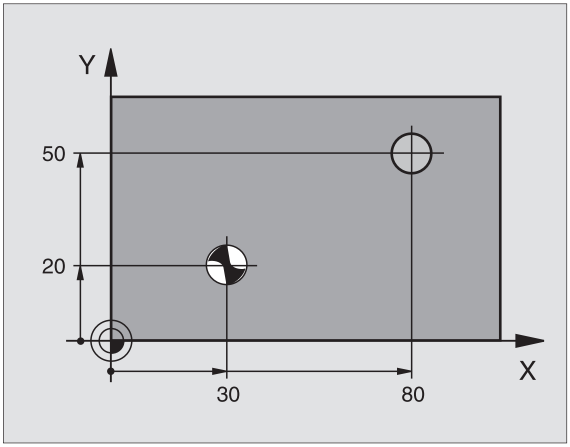
1. The TNC positions the tool in the tool axis at rapid traverse FMAX to the set-up clearance above the workpiece surface.
2. The tool is centered at the programmed feed rate F to the entered centering diameter or centering depth.
3. If defined, the tool remains at the centering depth.
4. Finally, the tool path is retraced to set-up clearance or—if programmed—to the 2nd set-up clearance at rapid traverse FMAX.



Before programming, note the following:

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

The algebraic sign for the cycle parameter Q344 (diameter) or Q201 (depth) determines the working direction. If you program the diameter or 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 diameter or depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

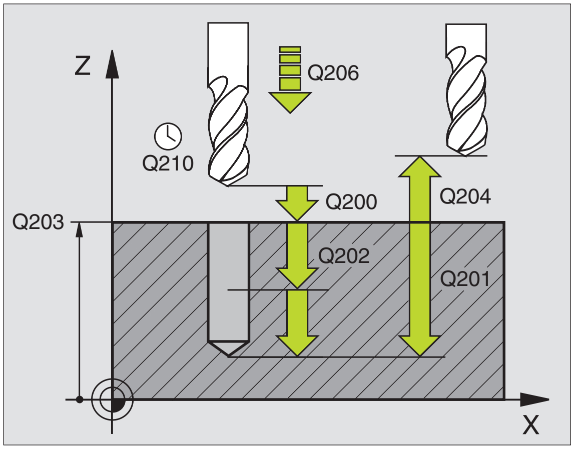
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Select Depth/Diameter (0/1)** Q343: Select whether centering is based on the entered diameter or depth. If centering is based on the entered diameter, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T.
- **Depth** Q201 (incremental value): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined.
- **Diameter (algebraic sign)** Q344: Centering diameter. Only effective if Q343=1 is defined.
- **Feed rate for plunging** Q206: Traversing speed of the tool during centering in mm/min.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- **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 G240 CENTERING
     Q200=2        ;SET-UP CLEARANCE
     Q343=1        ;SELECT DEPTH/DIA.
     Q201=+0       ;DEPTH
     Q344=-9       ;DIAMETER
     Q206=250      ;FEED RATE FOR PLUNGING
     Q211=0.1      ;DWELL TIME AT DEPTH
     Q203=+20      ;SURFACE COORDINATE
     Q204=100      ;2ND SET-UP CLEARANCE
N120 X+30 Y+20 M3 M99
N130 X+80 Y+50 M99
N140 Z+100 M2
```

DRILLING (Cycle G200)

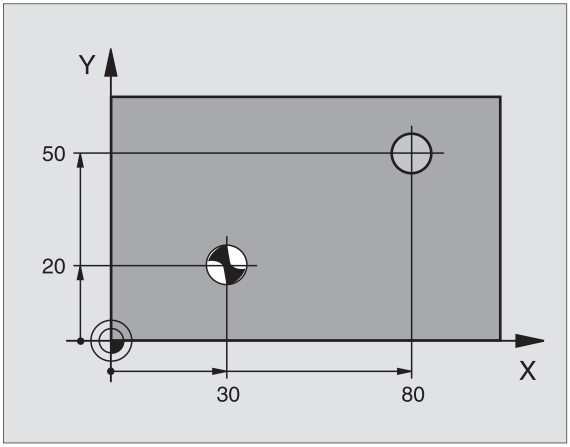
1 The TNC positions the tool in the tool axis at rapid traverse to the set-up clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate F.
3 The TNC returns the tool at rapid traverse to the setup clearance, dwells there (if a dwell time was entered), 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!

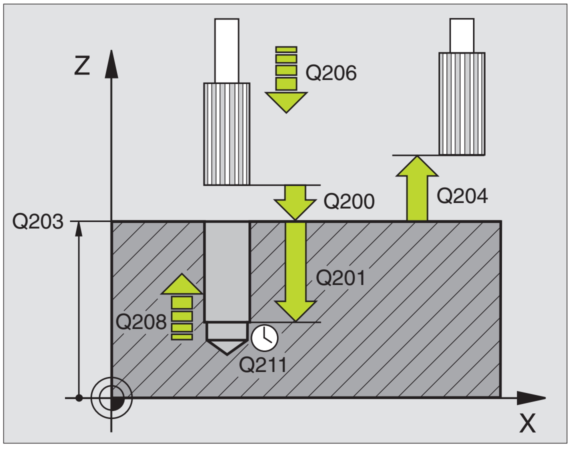
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- **Feed rate for plunging** Q206: Traversing speed of the tool during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

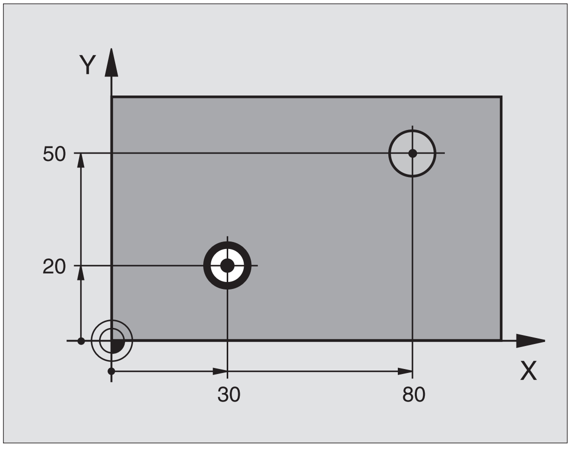
1. The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.
2. The tool reams to the entered depth at the programmed feed rate F.
3. If programmed, the tool remains at the hole bottom for the entered dwell time.
4. The tool then retracts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

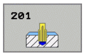
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



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.

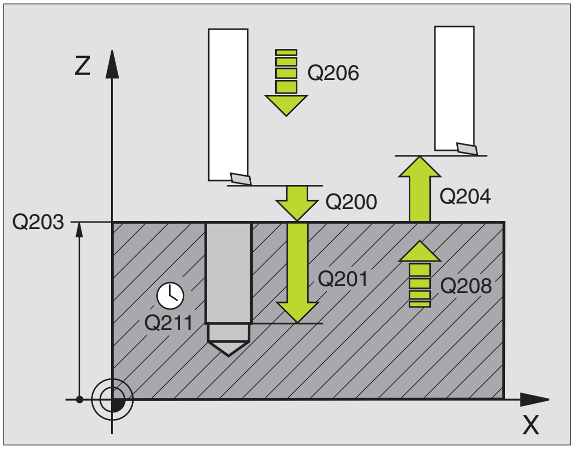
Example: NC blocks

```
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
```

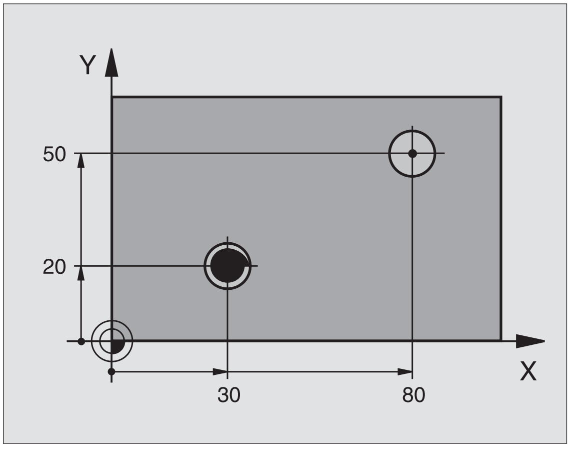
BORING (Cycle G202)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with controlled spindle.



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.**

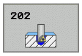
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

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.

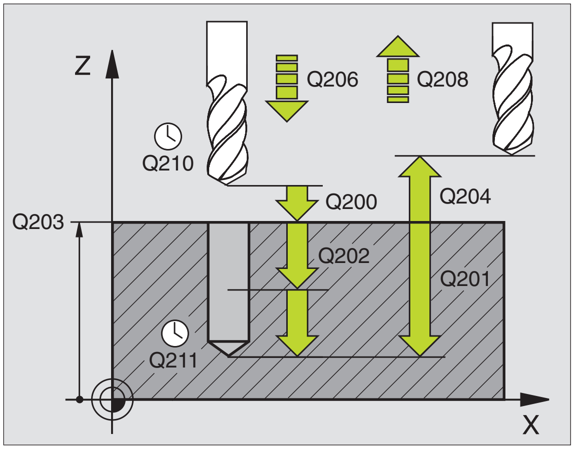
- **Angle for spindle orientation** Q336 (absolute value): Angle at which the TNC positions the tool before retracting it.

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 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.

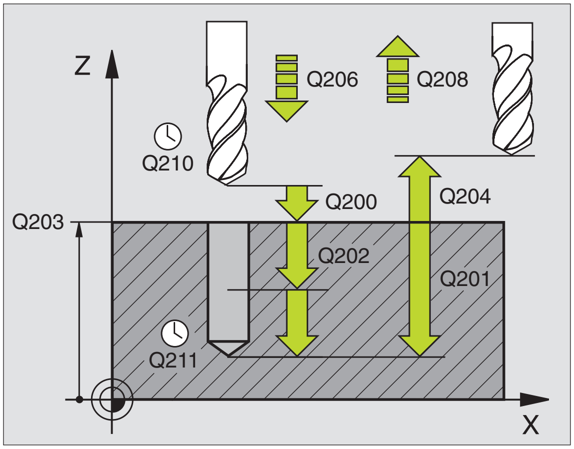
Danger of collision!

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!

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
```

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



Example: NC blocks

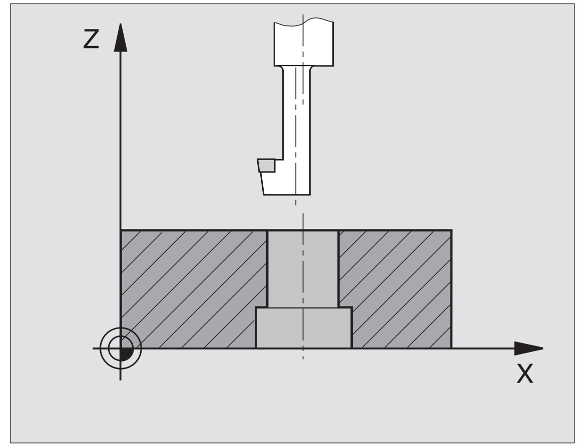
```
N110 G203 UNIVERSAL DRILLING
     Q200=2        ;SET-UP CLEARANCE
     Q201=-20      ;DEPTH
     Q206=150      ;FEED RATE FOR PLUNGING
     Q202=5        ;PLUNGING DEPTH
     Q210=0        ;DWELL TIME AT TOP
     Q203=+20      ;SURFACE COORDINATE
     Q204=50       ;2ND SET-UP CLEARANCE
     Q212=0.2      ;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
```

BACK BORING (Cycle G204)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

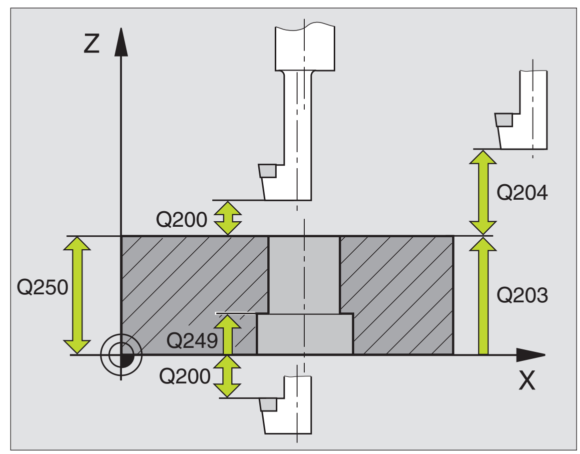
This cycle is effective only for machines with controlled spindle.

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.



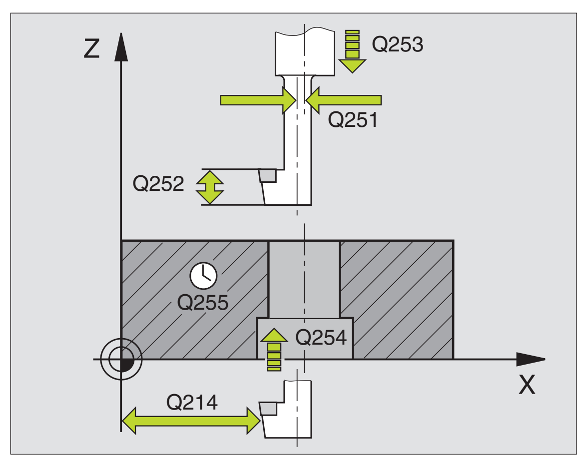
Before programming, note the following:

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 countersinking`** Q254: Traversing speed of the tool during countersinking in mm/min.
- **`Dwell time`** Q255: Dwell time in seconds at the top of the bore hole.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Disengaging direction (0/1/2/3/4)`** Q214: Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation).

  1 Retract tool in the negative ref. axis direction
  2 Retract tool in the neg. secondary axis direction
  3 Retract tool in the positive ref. axis direction
  4 Retract tool in the pos. secondary axis direction

Danger of collision!

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

- **`Angle for spindle orientation`** Q336 (absolute value): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole.

Example: NC blocks

```
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    ;FEED RATE FOR
                  COUNTERSINKING
     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 If you enter a deepened starting point, the TNC moves at the defined positioning feed rate to the set-up clearance above the deepened starting point.

3 The tool drills to the first plunging depth at the programmed feed rate F.

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

5 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.

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

7 The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to the set-up clearance at the retraction feed rate. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.

Before programming, note the following:

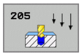
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

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.

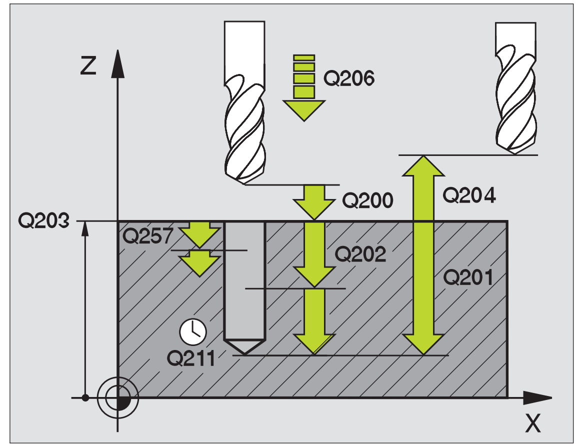
Danger of collision!

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.



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 ADVANCED 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
     Q379=7.5    ;STARTING POINT
     Q253=750    ;F PRE-POSITIONING
```

- **Infeed depth for chip breaking** Q257 (incremental value): Depth at which the TNC carries out chip breaking. There is no chip breaking if 0 is entered.
- **Retraction rate for chip breaking** Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position of drilling if a shorter tool has already pilot drilled to a certain depth. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance to the deepened starting point.
- **Feed rate for pre-positioning** Q253: Traversing velocity of the tool during positioning from the set-up clearance to a deepened starting point in mm/min. Effective only if Q379 is entered not equal to 0.

If you use Q379 to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. Retraction movements are not changed by the TNC, therefore they are calculated with respect to the coordinate of the workpiece surface.

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.

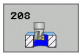
If you have entered the bore hole diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.

An active mirror function **does not** influence the type of milling defined in the 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!

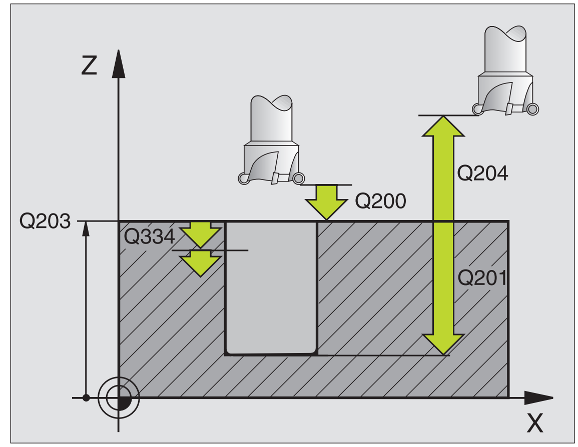


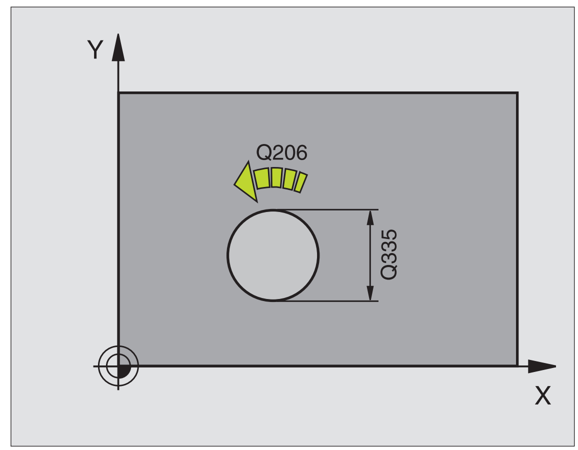
- **Set-up clearance** Q200 (incremental value): Distance between tool lower edge and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for plunging** Q206: Traversing speed of the tool during helical drilling in mm/min.
- **Infeed 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 181). 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.
- **Climb or up-cut** Q351: Type of milling operation with M3
  +1 = climb milling
  –1 = up-cut milling





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
     Q351=+1       ;CLIMB OR UP-CUT
```

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.

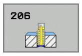
When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

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

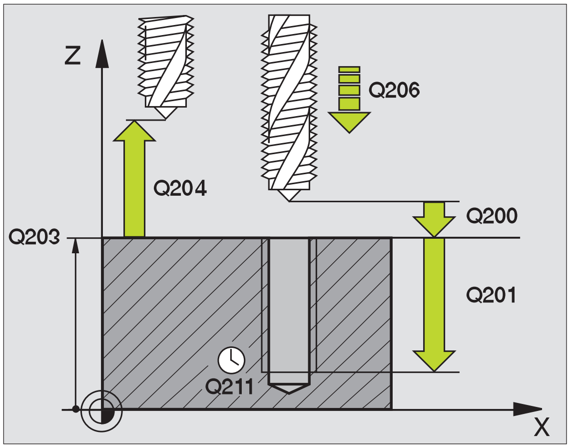
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

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.



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
```

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.

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.

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.

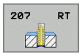
The feed-rate override knob is disabled.

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

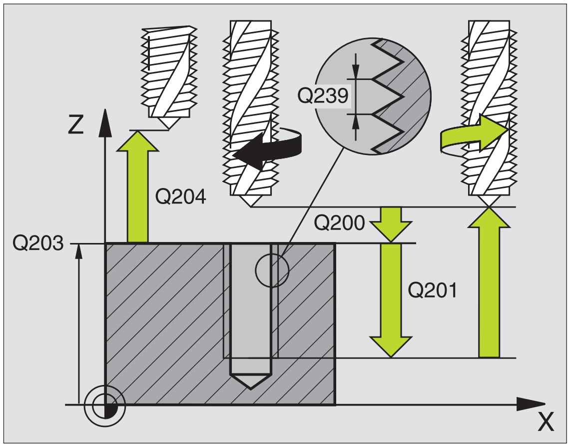
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

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

TAPPING WITH CHIP BREAKING (Cycle G209)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with controlled spindle.

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. If you have defined a factor for increasing the spindle speed, the TNC retracts from the hole at the corresponding speed
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.

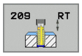
The feed-rate override knob is disabled.

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

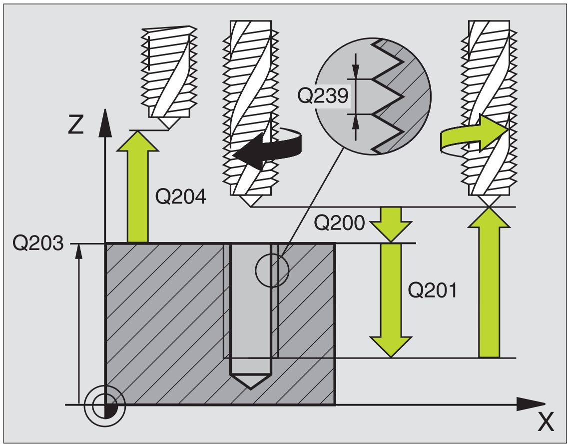
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

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 TAPPING W/ CHIP BRKG
    Q200=2      ;SET-UP CLEARANCE
    Q201=-20    ;DEPTH OF THREAD
    Q239=+1     ;PITCH
    Q203=+25    ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q257=5      ;DEPTH FOR CHIP BRKNG
    Q256=1      ;DIST. FOR CHIP BRKNG
    Q336=+0     ;ANGLE OF SPINDLE
```

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

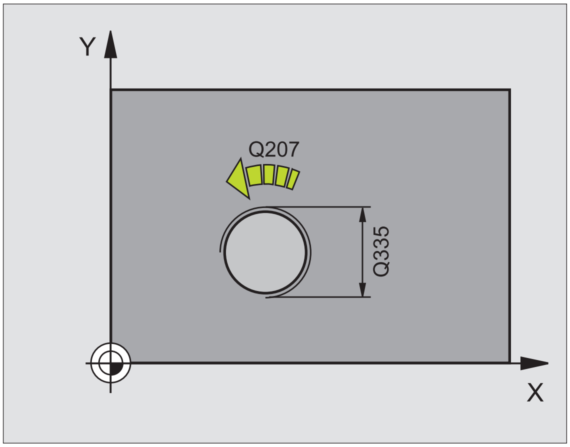
If a tool break occurs during thread cutting, stop the program run, change to the Positioning with MDI operating mode and move the tool in a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.

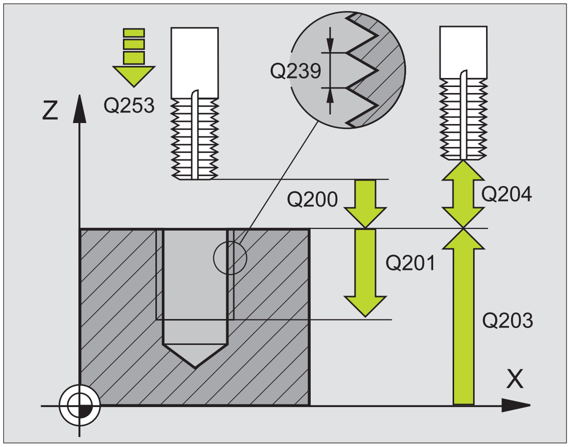
The TNC references the programmed feed rate during thread milling to the tool cutting edge. Since the TNC, however, always displays the feed rate relative to the path of the tool tip, the displayed value does not match the programmed value.

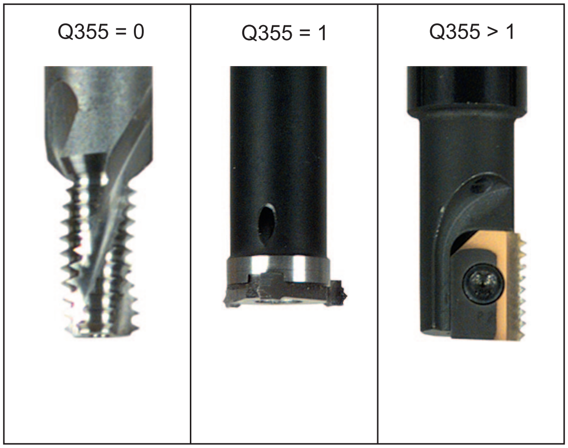
The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRRORING in only one axis.

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 in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.







Before programming, note the following:

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

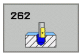
Note that the TNC makes a compensating movement in the tool axis before the approach movement. The length of the compensating motion depends on the thread pitch. Ensure sufficient space in the hole!

If you change the thread depth, the TNC automatically changes the starting point for the helical movement.

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 path to the depth of thread.
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **Climb or up-cut** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G262 THREAD MILLING
     Q335=10      ;NOMINAL DIAMETER
     Q239=+1.5    ;PITCH
     Q201=-20     ;DEPTH OF THREAD
     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 MILLNG
```

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 in a semicircle to the hole center.

Thread milling

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

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

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

11 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

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

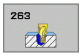
If you want to countersink with the front of the tool, define the countersinking depth as 0.

Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

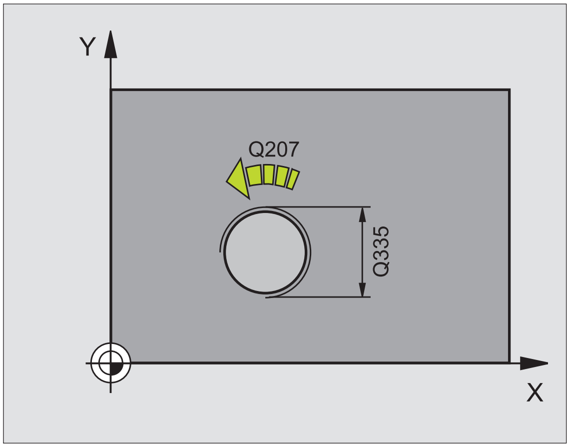
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

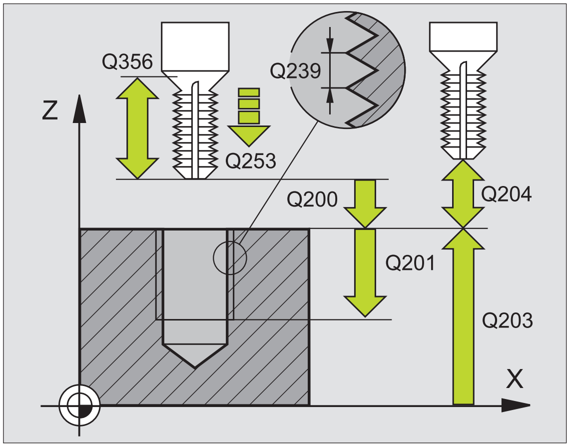
Danger of collision!

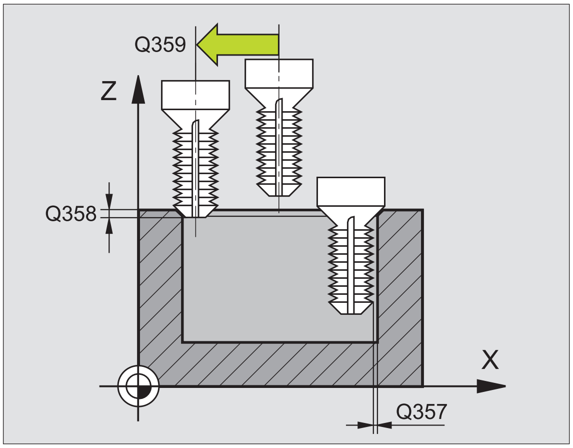
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **`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 countersinking`** Q254: Traversing speed of the tool during countersinking in mm/min.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G263 THREAD MLLNG/CNTSNKG
    Q335=10     ;NOMINAL DIAMETER
    Q239=+1.5   ;PITCH
    Q201=-16    ;DEPTH OF THREAD
    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    ;FEED RATE FOR
                 COUNTERSINKING
    Q207=500    ;FEED RATE FOR MILLNG
```

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 in a semicircle to the hole center.

Thread milling

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

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

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

12 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

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

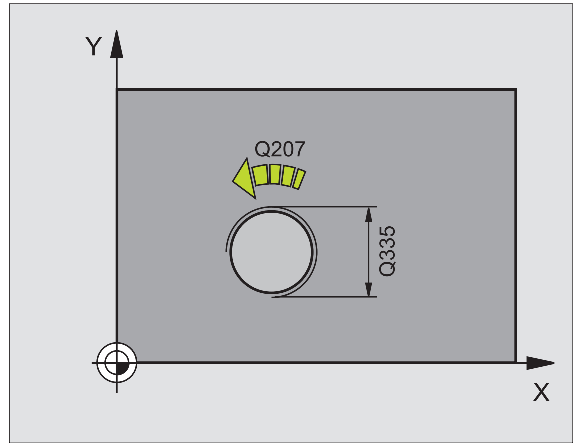
Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

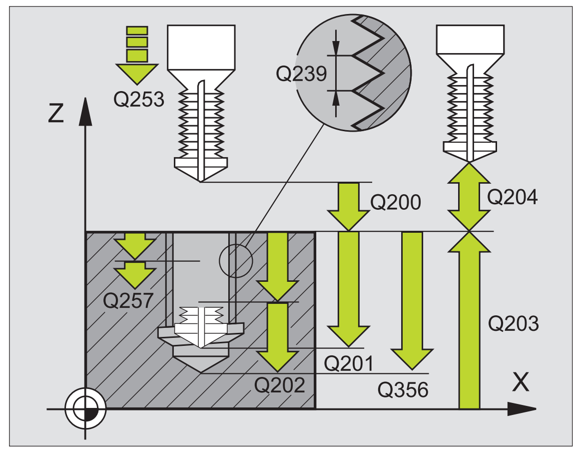
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

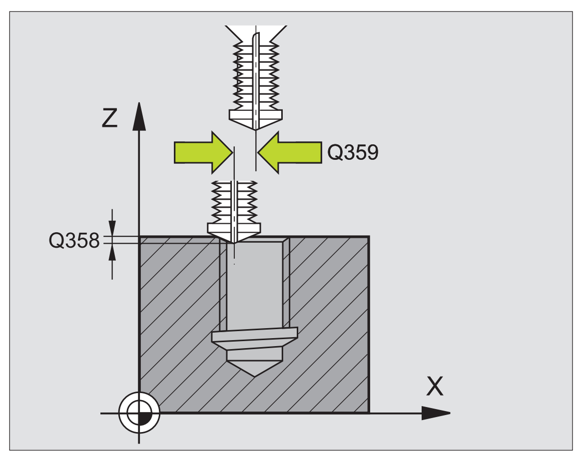
Danger of collision!

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     ;DEPTH OF THREAD
     Q356=-20     ;TOTAL HOLE DEPTH
     Q253=750     ;F PRE-POSITIONING
     Q351=+1      ;CLIMB OR UP-CUT
     Q202=5       ;PLUNGING DEPTH
     Q258=0.2     ;UPPER 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 MILLNG
```

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 in a semicircle to the hole center.

Thread milling

5 The tool moves at the programmed feed rate for pre-positioning to the starting plane for the thread.
6 The tool then approaches the thread diameter tangentially in a helical movement.
7 The tool moves on a continuous helical downward path until it reaches the thread depth.
8 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
9 At the end of the cycle, the TNC retracts the tool 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.

If you change the thread depth, the TNC automatically changes the starting point for the helical movement.

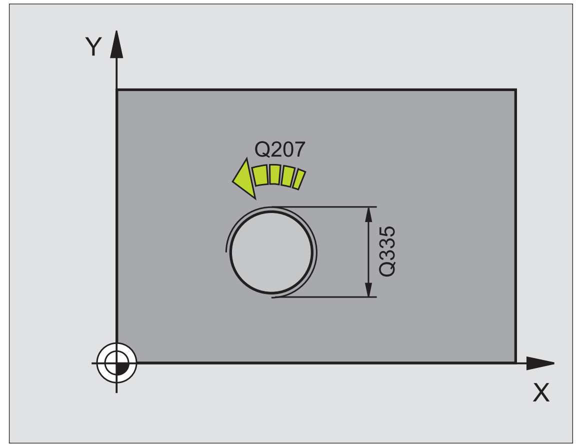
The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

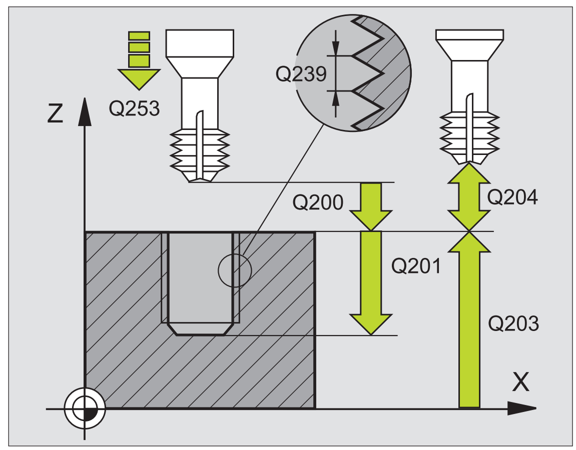
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

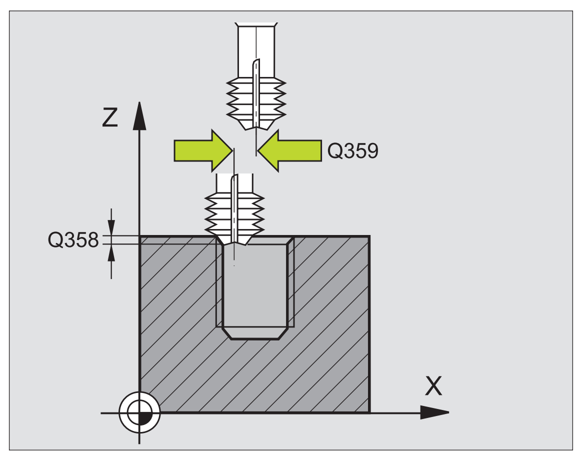
Danger of collision!

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 countersinking** Q254: Traversing speed of the tool during countersinking in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G265 HEL. THREAD DRLG/MLG
     Q335=10      ;NOMINAL DIAMETER
     Q239=+1.5    ;PITCH
     Q201=-16     ;DEPTH OF THREAD
     Q253=750     ;F PRE-POSITIONING
     Q358=+0      ;DEPTH AT FRONT
     Q359=+0      ;OFFSET AT FRONT
     Q360=0       ;COUNTERSINK
     Q200=2       ;SET-UP CLEARANCE
     Q203=+30     ;SURFACE COORDINATE
     Q204=50      ;2ND SET-UP CLEARANCE
     Q254=150     ;FEED RATE FOR
                   COUNTERSINKING
     Q207=500     ;FEED RATE FOR MILLNG
```

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

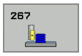
If you program a depth parameter to be 0, the TNC does not execute that step.

The algebraic sign for the cycle parameter thread depth determines the working direction.

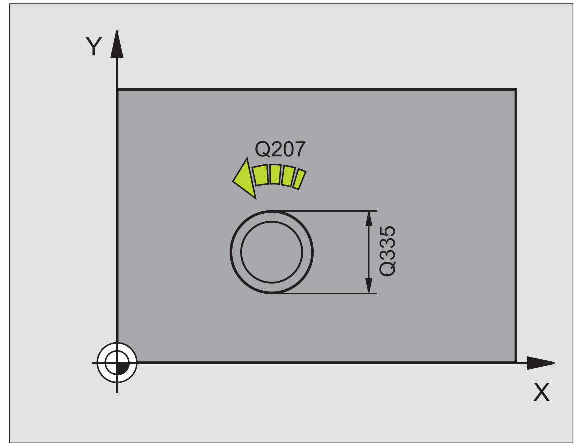
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

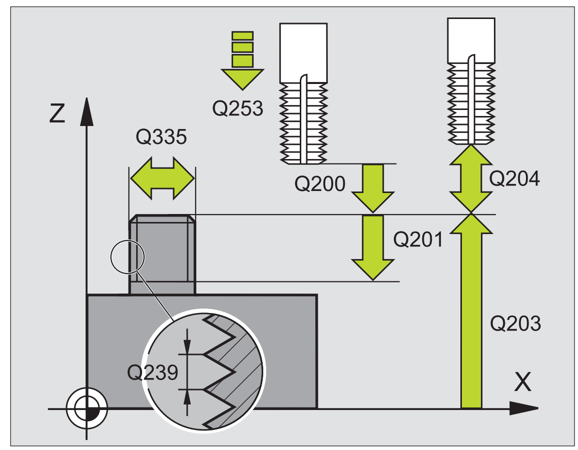
Danger of collision!

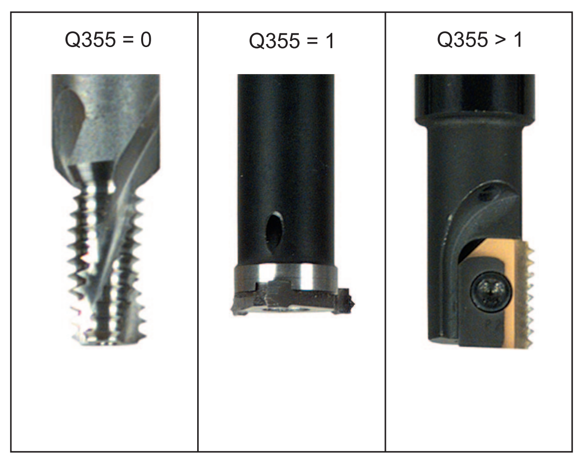
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **`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





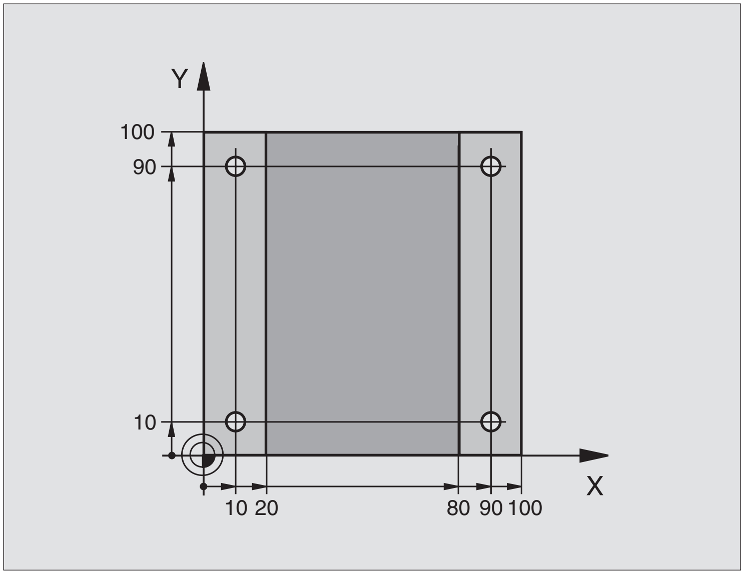


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth at front** Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- **Countersinking offset at front** Q359 (incremental 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 countersinking** Q254: Traversing speed of the tool during countersinking in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
N250 G267 OUTSIDE THREAD MLLNG
     Q335=10     ;NOMINAL DIAMETER
     Q239=+1.5   ;PITCH
     Q201=-20    ;DEPTH OF THREAD
     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    ;FEED RATE FOR
                  COUNTERSINKING
     Q207=500    ;FEED RATE FOR MILLNG
```

Example: Drilling cycles



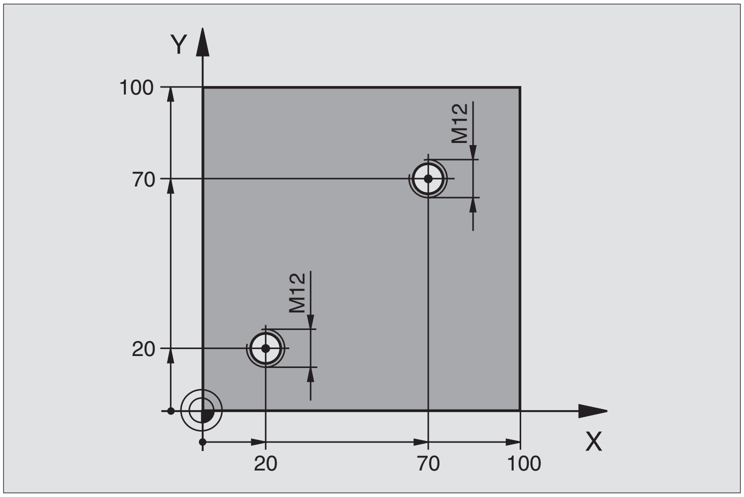
%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+3 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 DRILLING	Define cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR 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
`N99999999 %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 491)



`%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+3 *`	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
`N99999999 %C18 G71 *`	

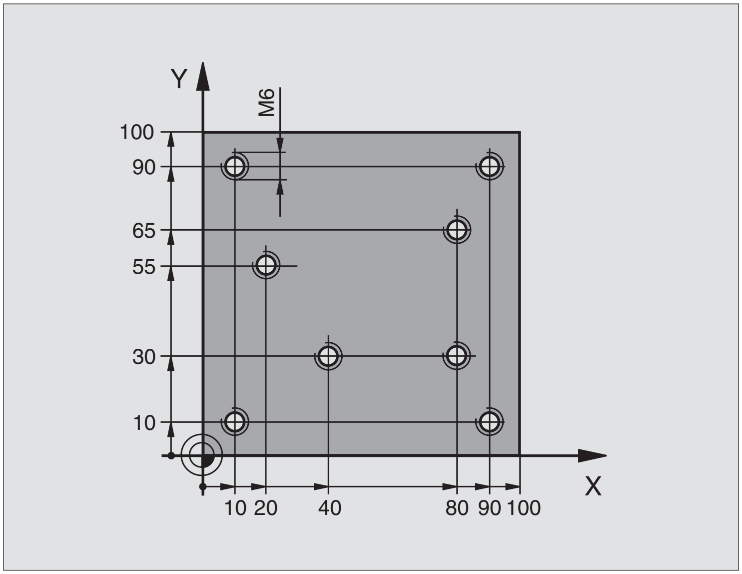
Example: Calling drilling cycles in connection with point tables

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 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+4 *`	Tool definition of center drill
`N40 G99 T2 L+0 R+2.4 *`	Define tool: drill
`N50 G99 T3 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`	

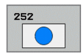
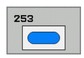
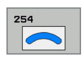
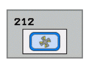
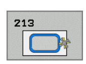
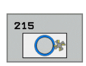
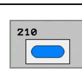
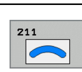
N100 G79 „PAT" F5000 M3 *	Cycle call in connection with point table TAB1.PNT
	Feed rate between points: 5000 mm/min
N110 G00 G40 Z+100 M6 *	Retract the tool, change the tool
N120 T2 G17 S5000 *	Call tool: drill
N130 G01 G40 Z+10 F5000 *	Move tool to clearance height (enter a value for F)
N140 G200 DRILLING	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 P03 0 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
N99999999 %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	Page
G251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging		Page 343
G252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging		Page 348
G253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal/ helical plunging		Page 352
G254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal/ helical plunging		Page 356
G212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance		Page 361
G213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance		Page 363
G214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance		Page 365
G215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance		Page 367
G210 SLOT RECIP. PLNG Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed		Page 369
G211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed		Page 371

RECTANGULAR POCKET (Cycle G251)

Use Cycle G251 RECTANGULAR POCKET to completely machine rectangular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

> With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Roughing

1. The tool plunges into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with Parameter Q366.
2. The TNC roughs out the pocket from the inside out, taking the overlap factor (Parameter Q370) and the finishing allowances (parameters Q368 and Q369) into account.
3. At the end of the roughing operation, the TNC moves the tool tangentially away from the pocket wall, then moves by the set-up clearance above the current pecking depth and returns from there at rapid traverse to the pocket center.
4. This process is repeated until the programmed pocket depth is reached.

Finishing

5 Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.

6 Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.

Before programming, note the following:

Pre-position the tool in the machining plane to the starting position with radius compensation R0. Note Parameter Q367 (pocket position).

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed **`G79:G01 X... Y`**... or in U and V if you programmed **`G79:G01 U... V...`**

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd set-up clearance).

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

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

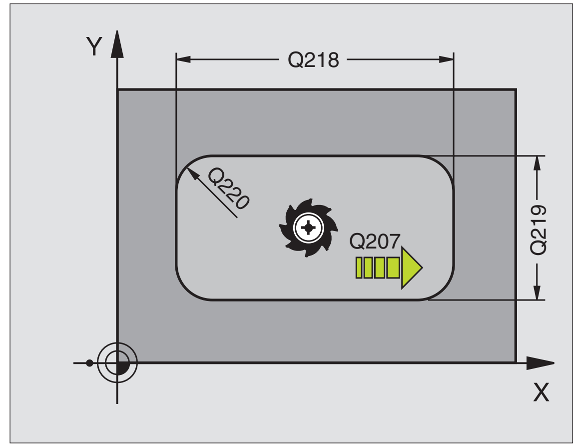
At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.

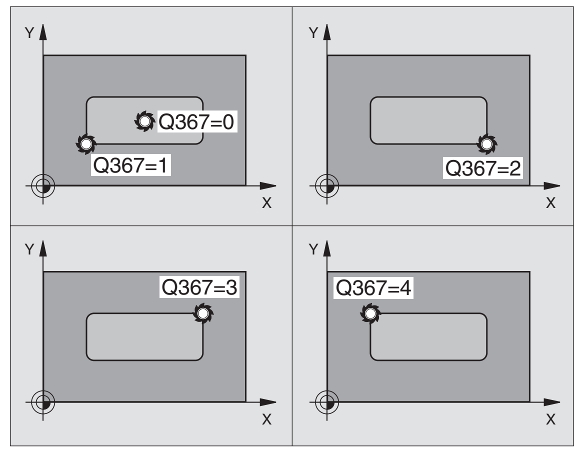
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

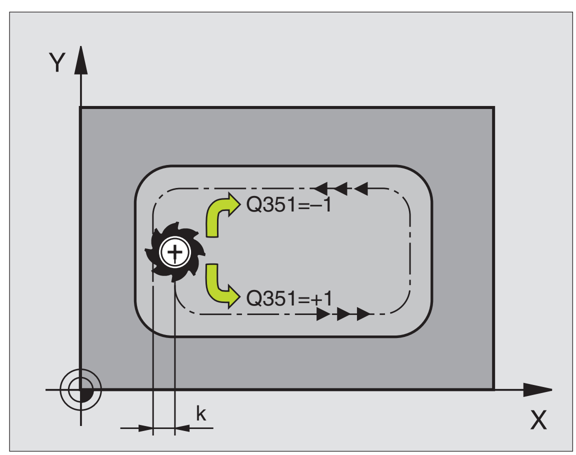
Danger of collision!

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!

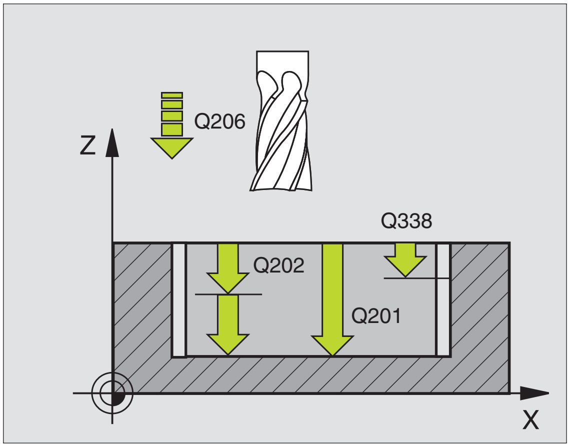
- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.
- **`First side length`** Q218 (incremental value): 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.
- **`Finishing allowance for side`** Q368 (incremental value): Finishing allowance in the working plane.
- **`Angle of rotation`** Q224 (absolute): Angle by which the entire pocket is rotated. The center of rotation is the position at which the tool is located when the cycle is called.
- **`Pocket position`** Q367: Position of the pocket in reference to the position of the tool when the cycle is called (see figure at center right):
  0: Tool position = Center of pocket
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.
- **`Climb or up-cut`** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling

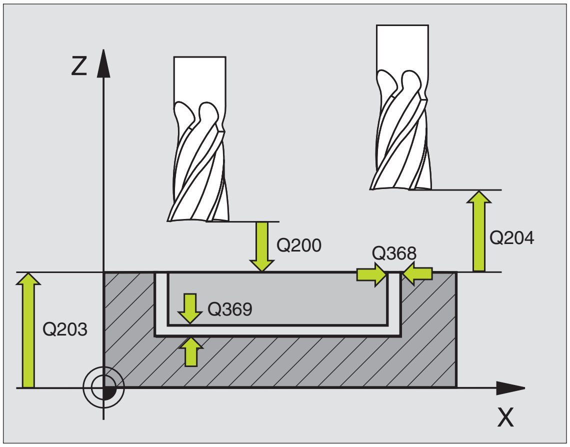






- **Depth** Q201 (incremental value): Distance between workpiece surface and pocket floor.
- **Plunging depth** Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis.
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min.
- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Workpiece surface coordinate** Q203 (absolute value): Absolute 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.





- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k.
- **Plunging strategy** Q366: Type of plunging strategy.
  - 0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1 = helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined not equal to 0. Otherwise the TNC displays an error message.
  - 2 = reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. The TNC will otherwise display an error message. The reciprocation length depends on the plunging angle. As a minimum value the TNC uses twice the tool diameter.
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min.

Example: NC blocks

```
N10 G251 RECTANGULAR POCKET
    Q215=0      ;MACHINING OPERATION
    Q218=80     ;FIRST SIDE LENGTH
    Q219=60     ;SECOND SIDE LENGTH
    Q220=5      ;CORNER RADIUS
    Q368=0.2    ;ALLOWANCE FOR SIDE
    Q224=+0     ;ANGLE OF ROTATION
    Q367=0      ;POCKET POSITION
    Q207=500    ;FEED RATE FOR MILLNG
    Q351=+1     ;CLIMB OR UP-CUT
    Q201=-20    ;DEPTH
    Q202=5      ;PLUNGING DEPTH
    Q369=0.1    ;ALLOWANCE FOR FLOOR
    Q206=150    ;FEED RATE FOR PLUNGING
    Q338=5      ;INFEED FOR FINISHING
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q370=1      ;TOOL PATH OVERLAP
    Q366=1      ;PLUNGE
    Q385=500    ;FEED RATE FOR FINISHING
N20 G79:G01 X+50 Y+50 Z+0 F15000 M3
```

CIRCULAR POCKET (Cycle G252)

Use Cycle 252 CIRCULAR POCKET to completely machine circular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

> With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Roughing

1. The tool plunges into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with Parameter Q366.
2. The TNC roughs out the pocket from the inside out, taking the overlap factor (Parameter Q370) and the finishing allowances (parameters Q368 and Q369) into account.
3. At the end of the roughing operation, the TNC moves the tool tangentially away from the pocket wall, then moves by the set-up clearance above the current pecking depth and returns from there at rapid traverse to the pocket center.
4. This process is repeated until the programmed pocket depth is reached.

Finishing

5 Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.

6 Then the TNC finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.

Before programming, note the following:

Pre-position the tool in the machining plane to the starting position (circle center) with radius compensation R0.

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed **G79:G01 X... Y**... or in U and V if you programmed **G79:G01 U... V...**

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd set-up clearance).

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

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

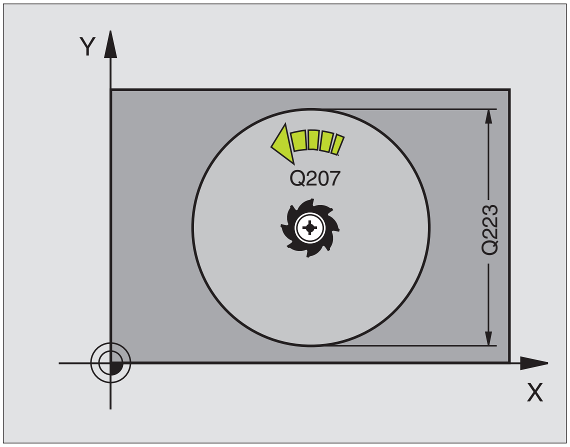
At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.

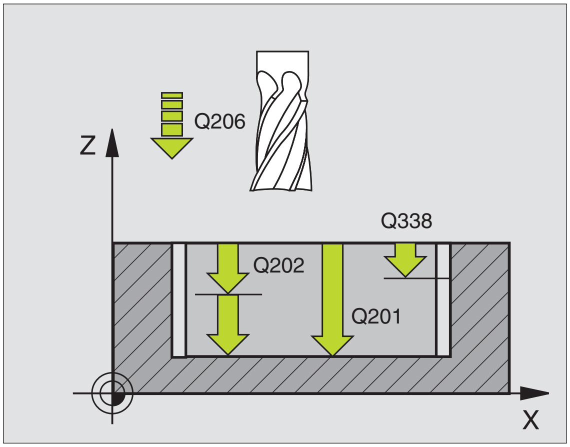
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

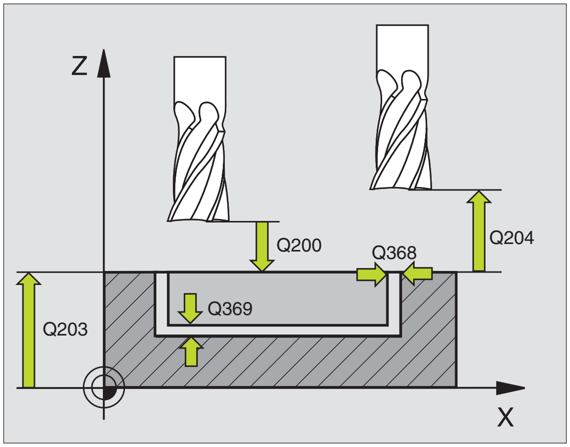
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.
- **`Circle diameter`** Q223: Diameter of the finished pocket.
- **`Finishing allowance for side`** Q368 (incremental value): Finishing allowance in the working plane.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.
- **`Climb or up-cut`** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and pocket floor.
- **`Plunging depth`** Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- **`Finishing allowance for floor`** Q369 (incremental value): Finishing allowance in the tool axis.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool while moving to depth in mm/min.
- **`Infeed for finishing`** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.





- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Workpiece surface coordinate** Q203 (absolute value): Absolute 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.
- **Path overlap factor** Q370: Q370 x tool radius = stepover factor k.
- **Plunging strategy** Q366: Type of plunging strategy.
  - 0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1 = helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined not equal to 0. Otherwise the TNC displays an error message.
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min.



Example: NC blocks

```
N10 G252 CIRCULAR POCKET
    Q215=0      ;MACHINING OPERATION
    Q223=60     ;CIRCLE DIAMETER
    Q368=0.2    ;ALLOWANCE FOR SIDE
    Q207=500    ;FEED RATE FOR MILLNG
    Q351=+1     ;CLIMB OR UP-CUT
    Q201=-20    ;DEPTH
    Q202=5      ;PLUNGING DEPTH
    Q369=0.1    ;ALLOWANCE FOR FLOOR
    Q206=150    ;FEED RATE FOR PLUNGING
    Q338=5      ;INFEED FOR FINISHING
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q370=1      ;TOOL PATH OVERLAP
    Q366=1      ;PLUNGE
    Q385=500    ;FEED RATE FOR FINISHING
N20 G79:G01 X+50 Y+50 Z+0 F15000 M3
```

SLOT MILLING (Cycle 253)

Use Cycle 253 to completely machine a slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

> With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Roughing

1. Starting from the left slot arc center, the tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with Parameter Q366.
2. The TNC roughs out the slot from the inside out, taking the finishing allowances (parameters Q368 and Q369) into account.
3. This process is repeated until the slot depth is reached.

Finishing

4 Inasmuch as finishing allowances are defined, the TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially in the right slot arc.

5 Then the TNC finishes the floor of the slot from the inside out. The slot floor is approached tangentially.

Before programming, note the following:

Pre-position the tool in the machining plane to the starting position with radius compensation R0. Note Parameter Q367 (slot position).

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed `G79:G01 X... Y`... or in U and V if you programmed `G79:G01 U... V...`

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd set-up clearance).

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

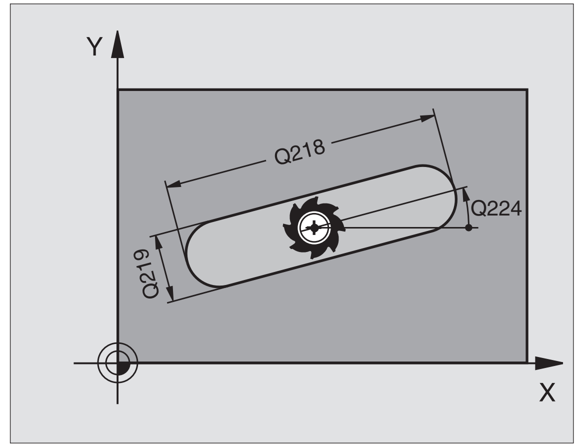
If the slot width is greater than twice the tool diameter, the TNC roughs the slot correspondingly from inside out. You can therefore mill any slots with small tools, too.

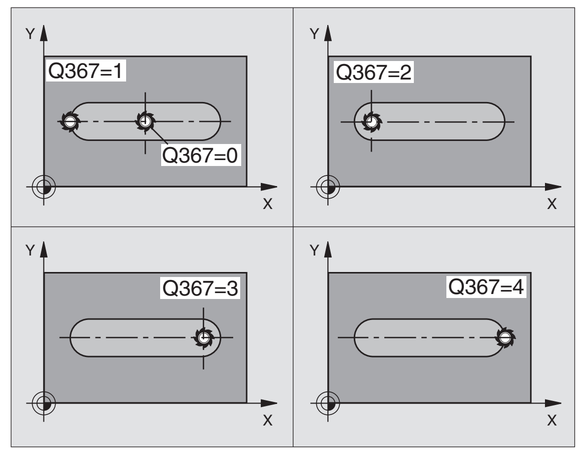
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

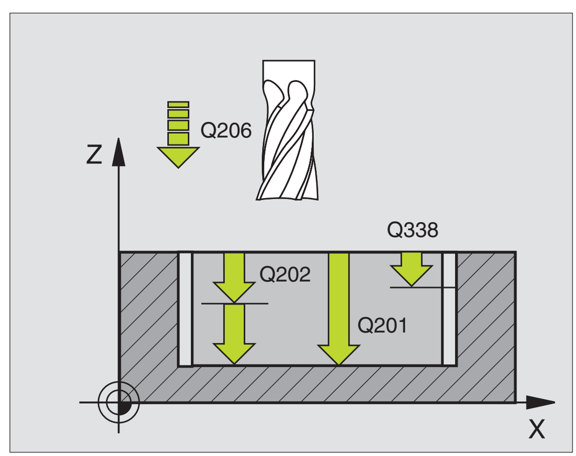
Danger of collision!

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!

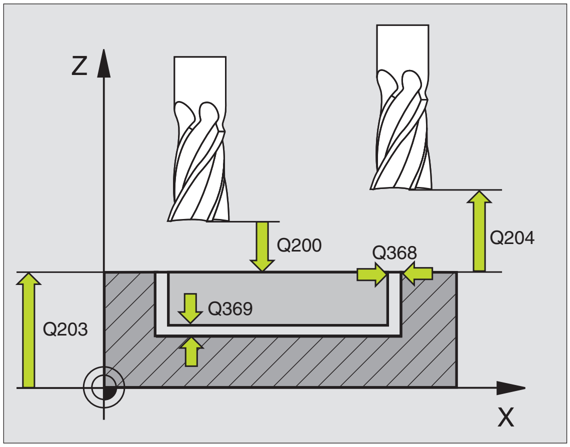
- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.
- **`Slot length`** Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot
- **`Slot width`** 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). Maximum slot width for roughing: Twice the tool diameter.
- **`Finishing allowance for side`** Q368 (incremental value): Finishing allowance in the working plane.
- **`Angle of rotation`** Q224 (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called.
- **`Slot position (0/1/2/3/4)`** Q367: Position of the slot in reference to the position of the tool when the cycle is called (see figure at center right):
  0: Tool position = Center of slot
  1: Tool position = Left end of slot
  2: Tool position = Center of left slot circle
  3: Tool position = Center of right slot circle
  4: Tool position = Right end of slot
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.
- **`Climb or up-cut`** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- **`Plunging depth`** Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- **`Finishing allowance for floor`** Q369 (incremental value): Finishing allowance in the tool axis.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool while moving to depth in mm/min.
- **`Infeed for finishing`** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.







- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Workpiece surface coordinate** Q203 (absolute value): Absolute 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.
- **Plunging strategy** Q366: Type of plunging strategy.
  - 0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1 = helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined not equal to 0. The TNC will otherwise display an error message. Plunge on a helical path only if there is enough space.
  - 2 = reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise the TNC displays an error message.
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min.



Example: NC blocks

```
N10 G253 SLOT MILLING
    Q215=0      ;MACHINING OPERATION
    Q218=80     ;SLOT LENGTH
    Q219=12     ;SLOT WIDTH
    Q368=0.2    ;ALLOWANCE FOR SIDE
    Q224=+0     ;ANGLE OF ROTATION
    Q367=0      ;SLOT POSITION
    Q207=500    ;FEED RATE FOR MILLNG
    Q351=+1     ;CLIMB OR UP-CUT
    Q201=-20    ;DEPTH
    Q202=5      ;PLUNGING DEPTH
    Q369=0.1    ;ALLOWANCE FOR FLOOR
    Q206=150    ;FEED RATE FOR PLUNGING
    Q338=5      ;INFEED FOR FINISHING
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q366=1      ;PLUNGE
    Q385=500    ;FEED RATE FOR FINISHING
N20 G79:G01 X+50 Y+50 Z+0 F15000 M3
```

CIRCULAR SLOT (Cycle 254)

Use Cycle 254 to completely machine a circular slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

> With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Roughing

1 The tool moves in a reciprocating motion in the slot center at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with Parameter Q366.

2 The TNC roughs out the slot from the inside out, taking the finishing allowances (parameters Q368 and Q369) into account.

3 This process is repeated until the slot depth is reached.

Finishing

4 Inasmuch as finishing allowances are defined, the TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially.

5 Then the TNC finishes the floor of the slot from the inside out. The slot floor is approached tangentially.

Before programming, note the following:

Pre-position the tool in the machining plane with radius compensation R0. Define Parameter Q367 **`(Reference for slot position)`** appropriately.

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed **`G79:G01 X... Y`**... or in U and V if you programmed **`G79:G01 U... V...`**

The TNC automatically pre-positions the tool in the tool axis. Note Parameter Q204 (2nd set-up clearance).

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

If the slot width is greater than twice the tool diameter, the TNC roughs the slot correspondingly from inside out. You can therefore mill any slots with small tools, too.

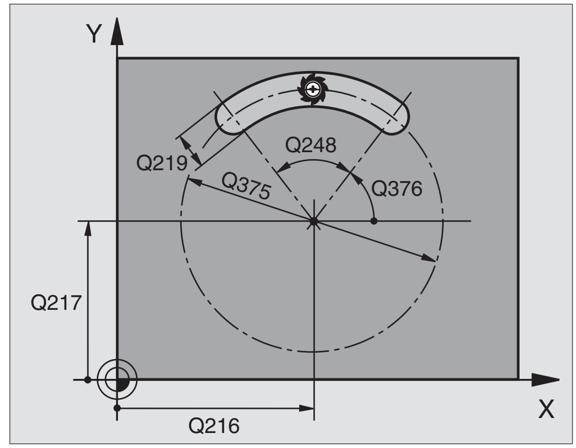
The slot position 0 is not allowed if you use Cycle G254 Circular Slot in together with Cycle G221.

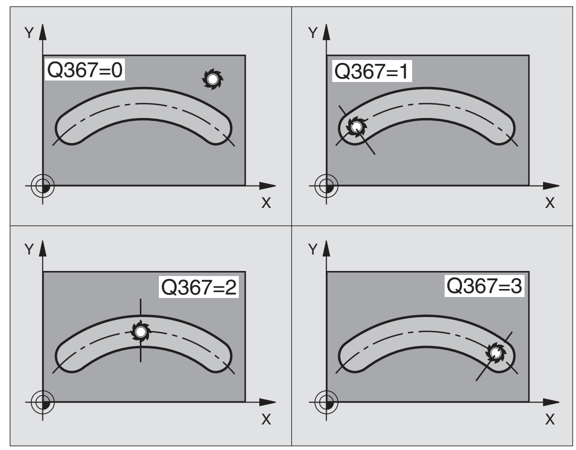
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

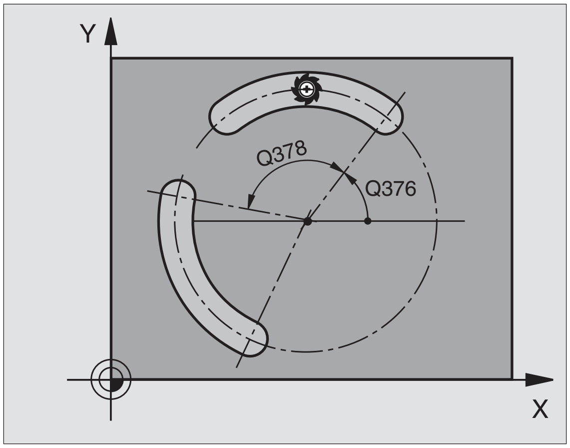
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

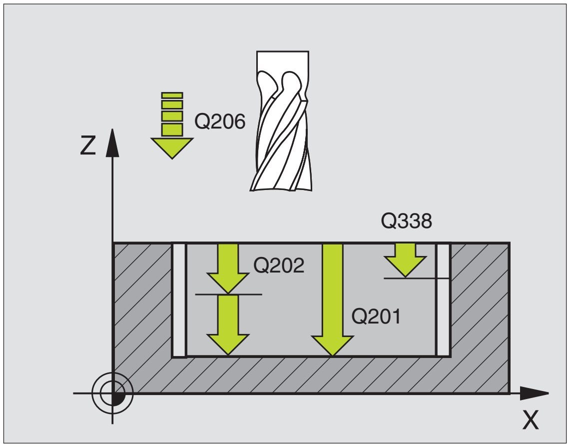
- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only executed if the finishing allowances (Q368, Q369) have been defined.
- **`Slot width`** Q219 (value parallel to the 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). Maximum slot width for roughing: Twice the tool diameter.
- **`Finishing allowance for side`** Q368 (incremental value): Finishing allowance in the working plane.
- **`Pitch circle diameter`** Q375: Enter the diameter of the pitch circle.
- **`Reference for slot position (0/1/2/3)`** Q367: Position of the slot in reference to the position of the tool when the cycle is called (see figure at center right):
  0: The tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle.
  1: Tool position = Center of left slot circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
  2: Tool position = Center of center line. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
  3: Tool position = Center of right slot circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
- **`Center in 1st axis`** Q216 (absolute value): Center of the pitch circle in the reference axis of the working plane. **Only effective if Q367 = 0.**
- **`Center in 2nd axis`** Q217 (absolute value): Center of the pitch circle in the minor axis of the working plane. **Only effective if Q367 = 0.**
- **`Starting angle`** Q376 (absolute value): Enter the polar angle of the starting point.
- **`Angular length`** Q248 (incremental value): Enter the angular length of the slot.



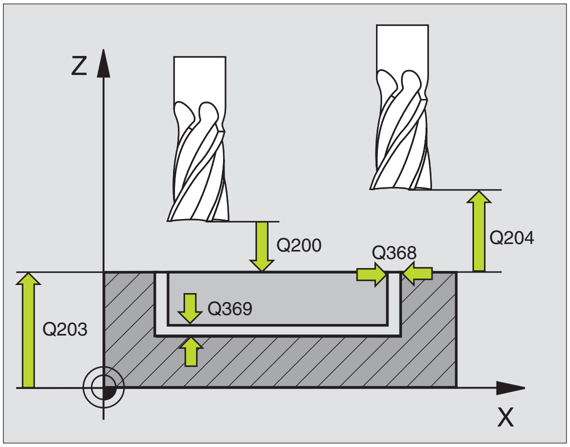


- **Angle increment** Q378 (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle.
- **Number of repetitions** Q377: Number of machining operations on the pitch circle.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Climb or up-cut** Q351: Type of milling operation with M03.
  +1 = climb milling
  –1 = up-cut milling
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- **Plunging depth** Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis.
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min.
- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.





- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Workpiece surface coordinate** Q203 (absolute value): Absolute 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.
- **Plunging strategy** Q366: Type of plunging strategy.
  - 0 = vertical plunging. The TNC plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table.
  - 1 = helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined not equal to 0. The TNC will otherwise display an error message. Plunge on a helical path only if there is enough space.
  - 2 = reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise the TNC displays an error message.
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min.

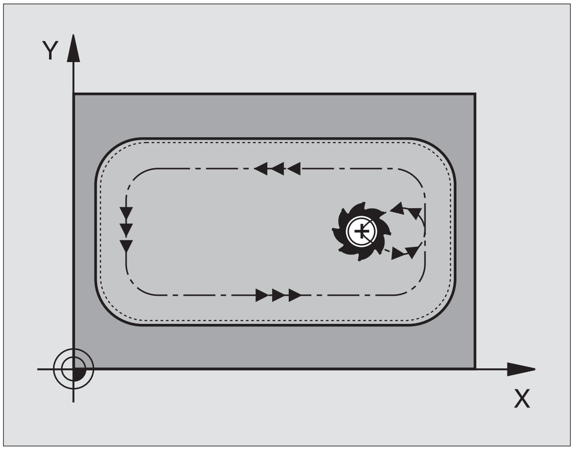


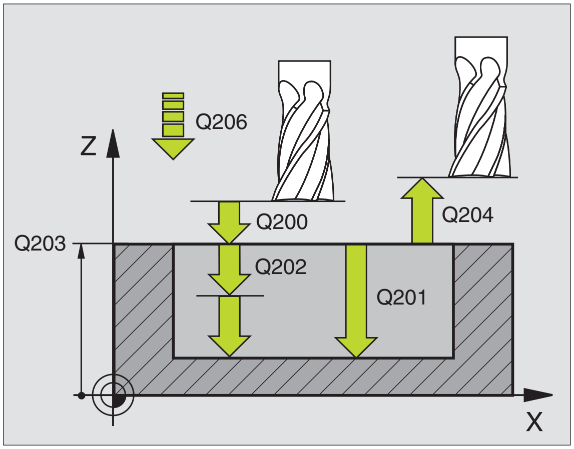
Example: NC blocks

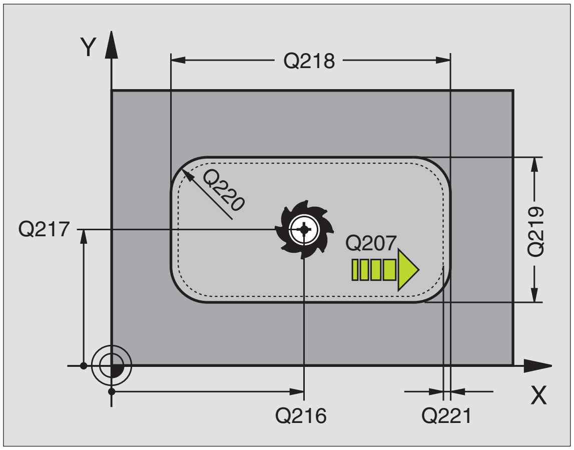
```
N10 G254 CIRCULAR SLOT
    Q215=0       ;MACHINING OPERATION
    Q219=12      ;SLOT WIDTH
    Q368=0.2     ;ALLOWANCE FOR SIDE
    Q375=80      ;PITCH CIRCLE DIA.
    Q367=0       ;REF. SLOT POSITION
    Q216=+50     ;CENTER 1ST AXIS
    Q217=+50     ;CENTER 2ND AXIS
    Q376=+45     ;STARTING ANGLE
    Q248=90      ;ANGULAR LENGTH
    Q378=0       ;STEPPING ANGLE
    Q377=1       ;NUMBER OF OPERATIONS
    Q207=500     ;FEED RATE FOR MILLNG
    Q351=+1      ;CLIMB OR UP-CUT
    Q201=-20     ;DEPTH
    Q202=5       ;PLUNGING DEPTH
    Q369=0.1     ;ALLOWANCE FOR FLOOR
    Q206=150     ;FEED RATE FOR PLUNGING
    Q338=5       ;INFEED FOR FINISHING
    Q200=2       ;SET-UP CLEARANCE
    Q203=+0      ;SURFACE COORDINATE
    Q204=50      ;2ND SET-UP CLEARANCE
    Q366=1       ;PLUNGE
    Q385=500     ;FEED RATE FOR FINISHING
N20 G79:G01 X+50 Y+50 Z+0 F15000 M3
```

POCKET FINISHING (Cycle G212)

1 The TNC M automatically moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.
2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the allowance and tool radius into account for calculating the starting point. If necessary, the TNC penetrates at the pocket center.
3 If the tool is at the 2nd set-up clearance, it moves at rapid traverse 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!

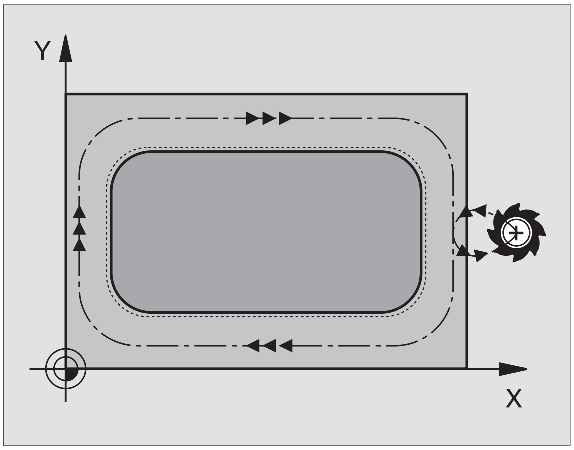
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min when moving to depth. If you 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.

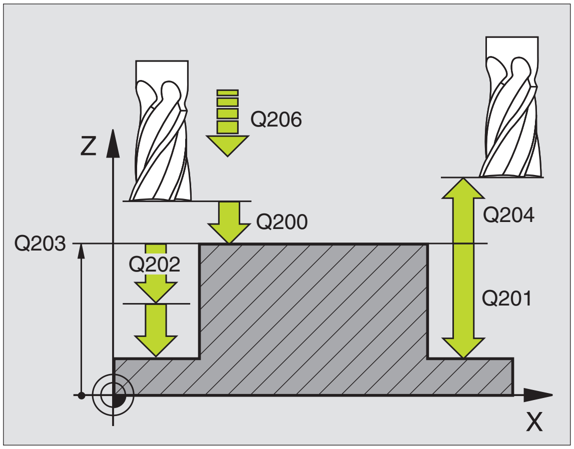
Example: NC blocks

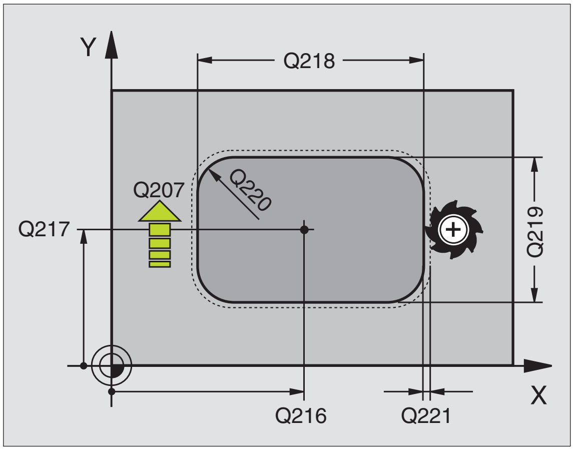
```
N350 G212 POCKET FINISHING
     Q200=2        ;SET-UP CLEARANCE
     Q201=-20      ;DEPTH
     Q206=150      ;FEED RATE FOR PLUNGING
     Q202=5        ;PLUNGING DEPTH
     Q207=500      ;FEED RATE FOR MILLNG
     Q203=+30      ;SURFACE COORDINATE
     Q204=50       ;2ND SET-UP CLEARANCE
     Q216=+50      ;CENTER 1ST AXIS
     Q217=+50      ;CENTER 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.

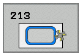
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

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!



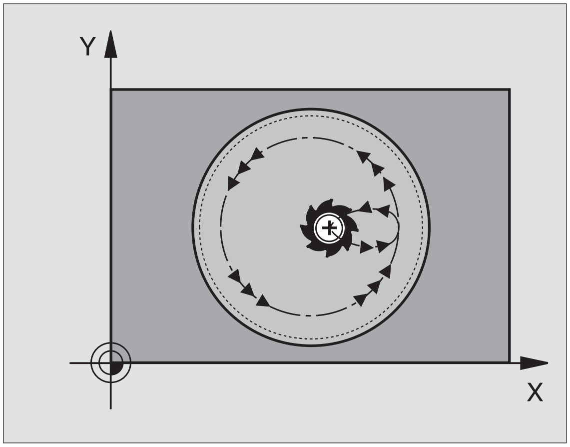
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting 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 MILLNG
     Q203=+30    ;SURFACE COORDINATE
     Q294=50     ;2ND SET-UP CLEARANCE
     Q216=+50    ;CENTER 1ST AXIS
     Q217=+50    ;CENTER 2ND AXIS
     Q218=80     ;FIRST SIDE LENGTH
     Q219=60     ;SECOND SIDE LENGTH
     Q220=5      ;CORNER RADIUS
     Q221=0      ;OVERSIZE
```

CIRCULAR POCKET FINISHING (Cycle G214)

1 The TNC M automatically moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.
2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the workpiece blank diameter and tool radius into account for calculating the starting point. If you enter a workpiece blank diameter of 0, the TNC plunge-cuts into the pocket center.
3 If the tool is at the 2nd set-up clearance, it moves 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).

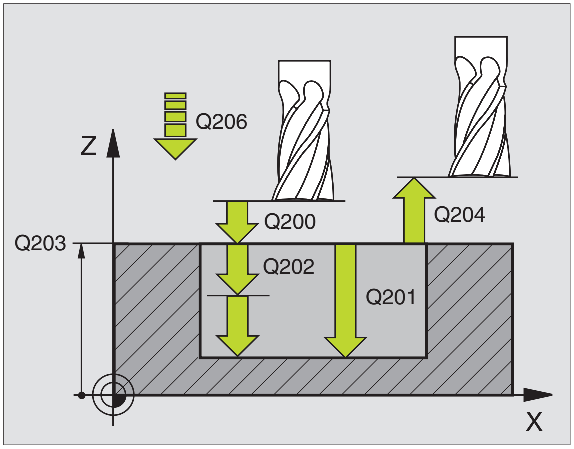


Before programming, note the following:

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.

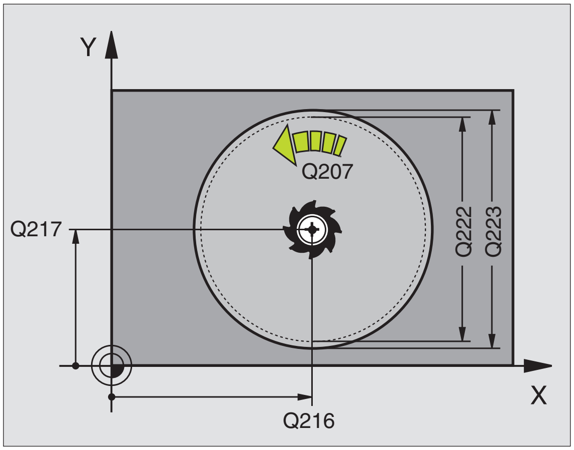
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



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 C. POCKET FINISHING
     Q200=2       ;SET-UP CLEARANCE
     Q201=-20     ;DEPTH
     Q206=150     ;FEED RATE FOR PLUNGING
     Q202=5       ;PLUNGING DEPTH
     Q207=500     ;FEED RATE FOR MILLNG
     Q203=+30     ;SURFACE COORDINATE
     Q204=50      ;2ND SET-UP CLEARANCE
     Q216=+50     ;CENTER 1ST AXIS
     Q217=+50     ;CENTER 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 pocket.
2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud at a distance of approx. twice the tool radius.
3 If the tool is at the 2nd set-up clearance, it moves 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.

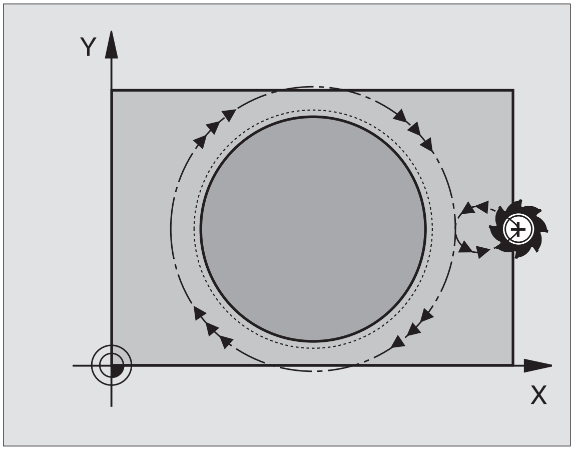
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

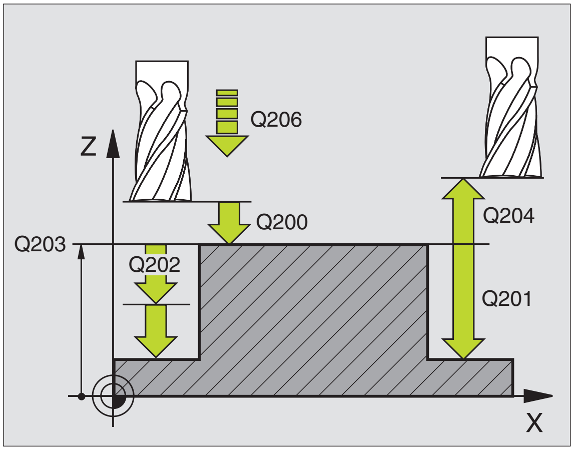
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

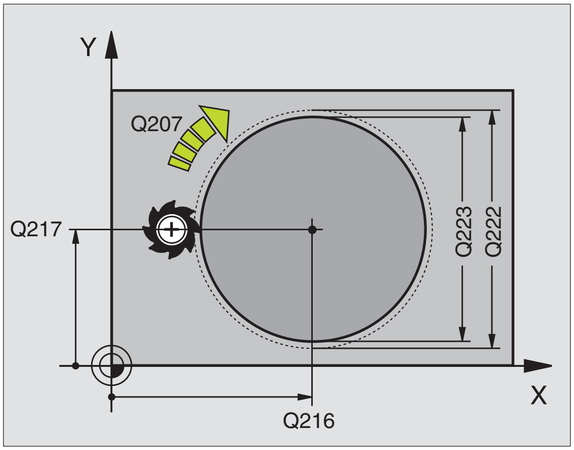
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

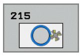
Danger of collision!

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.

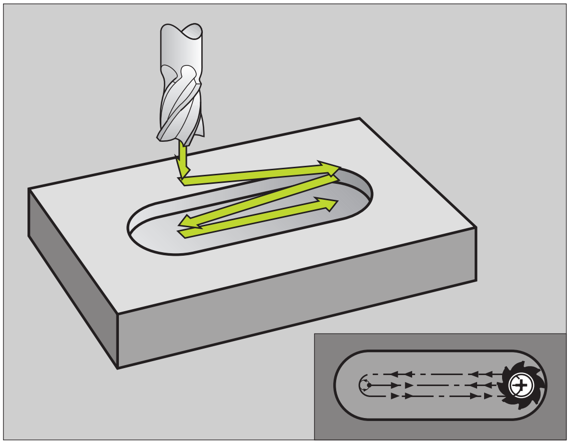
Example: NC blocks

```
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 MILLNG
     Q203=+30     ;SURFACE COORDINATE
     Q204=50      ;2ND SET-UP CLEARANCE
     Q216=+50     ;CENTER 1ST AXIS
     Q217=+50     ;CENTER 2ND AXIS
     Q222=81      ;WORKPIECE BLANK DIA.
     Q223=80      ;FINISHED PART DIA.
```

SLOT with reciprocating plunge-cut (Cycle G210)

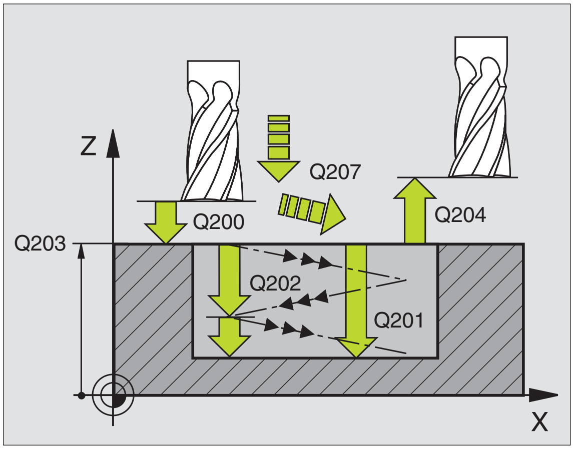
Roughing

1. At rapid traverse, the TNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the left circle. From there, the TNC positions the tool to the set-up clearance above the workpiece surface.
2. The tool moves at the feed rate for milling to the workpiece surface. From there, the cutter advances in the longitudinal direction of the slot—plunge-cutting obliquely into the material—until it reaches the center of the right circle.
3. The tool then moves back to the center of the left circle, again with oblique plunge-cutting. This process is repeated until the programmed milling depth is reached.
4. 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

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:

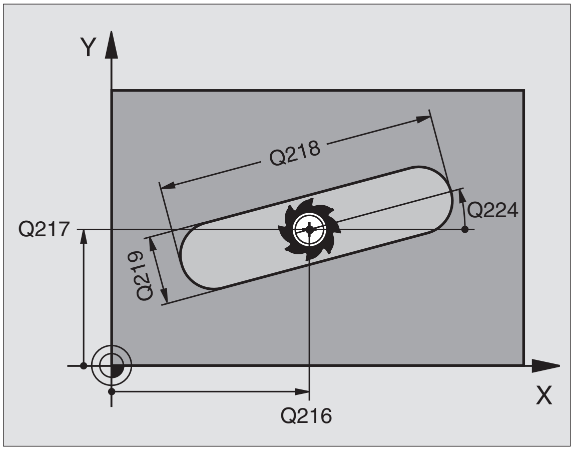
The TNC automatically pre-positions the tool in the tool axis and working plane.

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 MILLNG
     Q202=5      ;PLUNGING DEPTH
     Q215=0      ;MACHINING OPERATION
     Q203=+30    ;SURFACE COORDINATE
     Q204=50     ;2ND SET-UP CLEARANCE
     Q216=+50    ;CENTER 1ST AXIS
     Q217=+50    ;CENTER 2ND AXIS
     Q218=80     ;FIRST SIDE LENGTH
     Q219=12     ;SECOND SIDE LENGTH
     Q224=+15    ;ROTATIONAL POSITION
     Q338=5      ;INFEED FOR FINISHING
     Q206=150    ;FEED RATE FOR PLUNGING
```

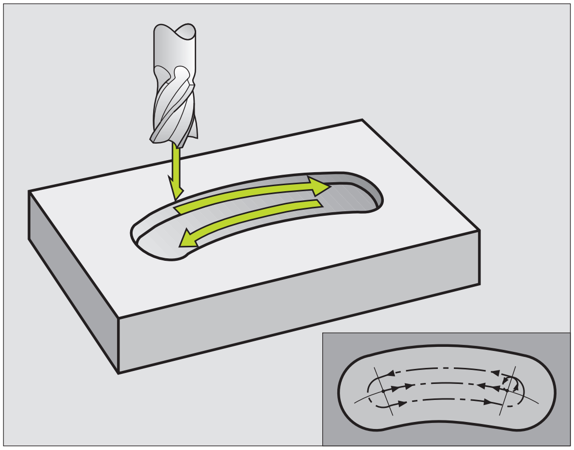
CIRCULAR SLOT with reciprocating plunge-cut (Cycle G211)

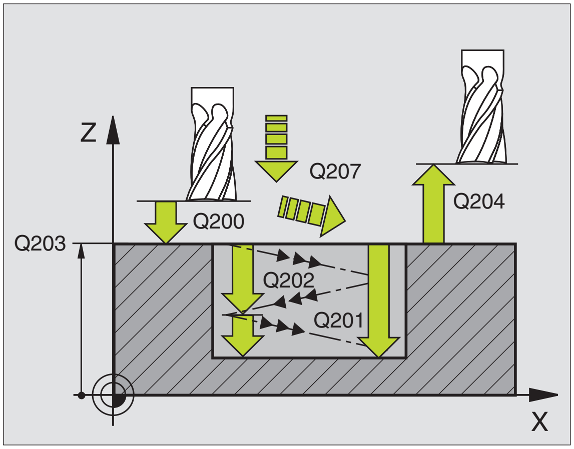
Roughing

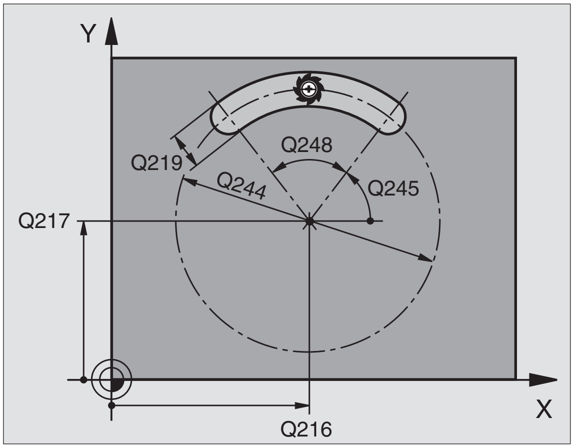
1 At rapid traverse, the TNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the right circle. From there, the tool is positioned to the programmed set-up clearance above the workpiece surface.
2 The tool moves at the milling feed rate to the workpiece surface. From there, the cutter advances—plunge-cutting obliquely into the material—to the other end of the slot.
3 The tool then moves at a downward angle back to the starting point, again with oblique plunge-cutting. This process (steps 2 to 3) is repeated until the programmed milling depth is reached.
4 For the purpose of face milling, the TNC moves the tool at the milling depth to the other end of the slot.

Finishing

5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed. The starting point for the finishing process is the center of the right circle.
6 When the tool reaches the end of the contour, it departs the contour tangentially.
7 At the end of the cycle, the tool is retracted at rapid traverse 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!

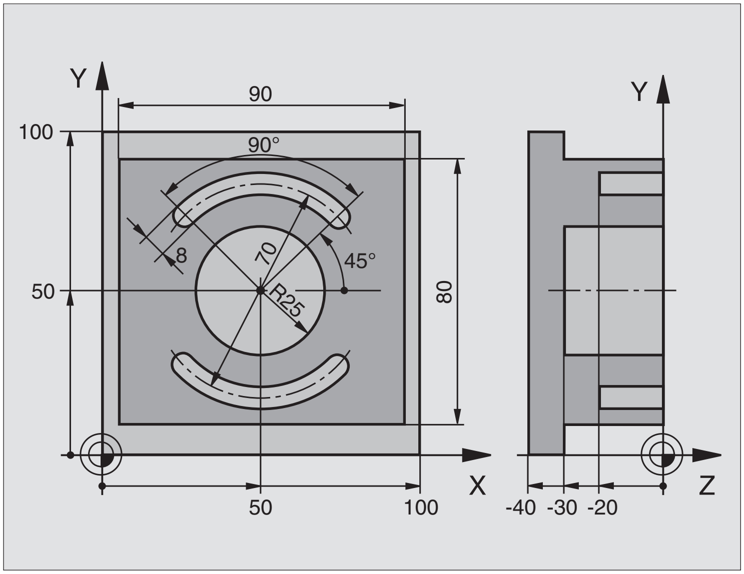
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **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 MILLNG
     Q202=5      ;PLUNGING DEPTH
     Q215=0      ;MACHINING OPERATION
     Q203=+30    ;SURFACE COORDINATE
     Q204=50     ;2ND SET-UP CLEARANCE
     Q216=+50    ;CENTER 1ST AXIS
     Q217=+50    ;CENTER 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 1ST AXIS	
Q217=+50 ;CENTER 2ND AXIS	
Q218=90 ;FIRST SIDE LENGTH	
Q219=80 ;SECOND SIDE LENGTH	
Q220=0 ;CORNER RADIUS	
Q221=5 ;OVERSIZE	

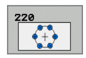
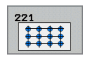
`N80 G79 M03 *`	Call cycle for machining the contour outside
`N90 G252 CIRCULAR POCKET`	Define CIRCULAR POCKET MILLING cycle
`Q215=0 ;MACHINING OPERATION`	
`Q223=50 ;CIRCLE DIAMETER`	
`Q368=0.2 ;ALLOWANCE FOR SIDE`	
`Q207=500 ;FEED RATE FOR MILLNG`	
`Q351=+1 ;CLIMB OR UP-CUT`	
`Q201=-30 ;DEPTH`	
`Q202=5 ;PLUNGING DEPTH`	
`Q369=0.1 ;ALLOWANCE FOR FLOOR`	
`Q206=150 ;FEED RATE FOR PLUNGING`	
`Q338=5 ;INFEED FOR FINISHING`	
`Q200=2 ;SET-UP CLEARANCE`	
`Q203=+0 ;SURFACE COORDINATE`	
`Q204=50 ;2ND SET-UP CLEARANCE`	
`Q370=1 ;TOOL PATH OVERLAP`	
`Q366=1 ;PLUNGE`	
`Q385=750 ;FEED RATE FOR FINISHING`	
`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 G254 CIRCULAR SLOT`	Define SLOT cycle
`Q215=0 ;MACHINING OPERATION`	
`Q219=8 ;SLOT WIDTH`	
`Q368=0.2 ;ALLOWANCE FOR SIDE`	
`Q375=70 ;PITCH CIRCLE DIA.`	
`Q367=0 ;REF. SLOT POSITION`	No pre-positioning in X/Y required
`Q216=+50 ;CENTER 1ST AXIS`	
`Q217=+50 ;CENTER 2ND AXIS`	
`Q376=+45 ;STARTING ANGLE`	
`Q248=90 ;ANGULAR LENGTH`	
`Q378=180 ;STEPPING ANGLE`	Starting point for second slot
`Q377=2 ;NUMBER OF OPERATIONS`	
`Q207=500 ;FEED RATE FOR MILLNG`	
`Q351=+1 ;CLIMB OR UP-CUT`	
`Q201=-20 ;DEPTH`	
`Q202=5 ;PLUNGING DEPTH`	

Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLUNGING	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q366=1 ;PLUNGE	
Q385=750 ;FEED RATE FOR FINISHING	
N150 G01 X+50 Y+50 F10000 M03 G79 *	Call SLOT cycle
N160 G00 Z+250 M02 *	Retract in the tool axis, end program
N99999999 %C210 G71 *	

8.5 Cycles for Machining Point Patterns

Overview

The TNC provides two cycles for machining point patterns directly:

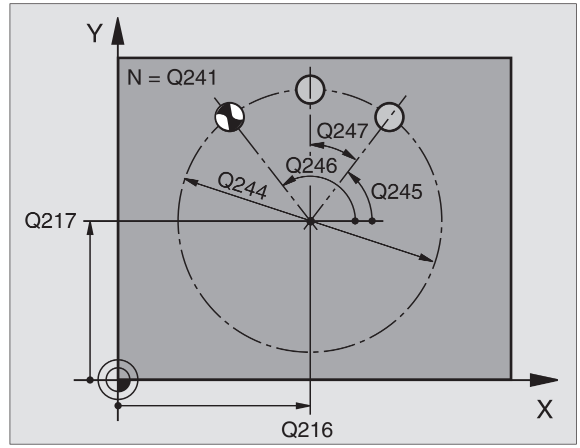
Cycle	Soft key	Page
G220 CIRCULAR PATTERN		Page 377
G221 LINEAR PATTERN		Page 379

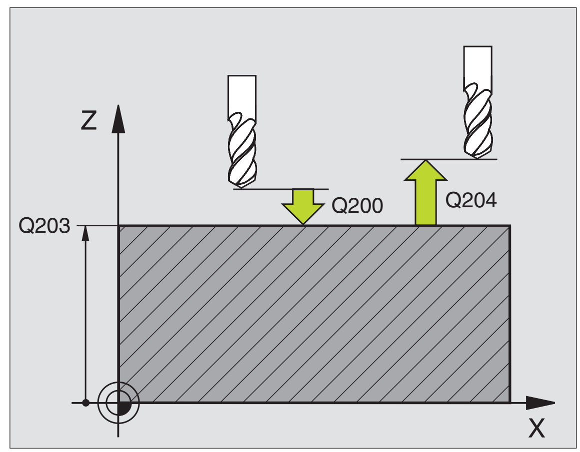
You can combine Cycle G220 and Cycle G221 with the following fixed cycles:

> If you have to machine irregular point patterns, use **G79 "PAT"** to develop point tables (see "Point Tables" on page 288).

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 G240 CENTERING
Cycle G251 RECTANGULAR POCKET
Cycle G252 CIRCULAR POCKET MILLING
Cycle G253 SLOT MILLING
Cycle G254 CIRCULAR SLOT (cannot be combined with Cycle 220)
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:

- Move to 2nd setup clearance (spindle)
- 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.

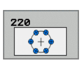
3 The tool then approaches the starting point for the next machining operation on a straight line at set-up clearance (or 2nd set-up clearance).

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 counterclockwise; otherwise, machining will be clockwise.

- **Stepping angle** Q247 (incremental value): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the angle step determines the working direction (– = clockwise).
- **Number of repetitions** Q241: Number of machining operations on a pitch circle.
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Moving to clearance height** Q301: Definition of how the tool is to move between machining processes.
  0: Move to the set-up clearance between operations.
  1: Move to the 2nd set-up clearance between the measuring points.
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between machining operations.
  0: Move between operations on a straight line
  1: Move between operations on the pitch circle

Example: NC blocks

```
N530 G220 POLAR PATTERN
     Q216=+50   ;CENTER 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     ;NUMBER OF OPERATIONS
     Q200=2     ;SET-UP CLEARANCE
     Q203=+30   ;SURFACE COORDINATE
     Q204=50    ;2ND SET-UP CLEARANCE
     Q203=1     ;MOVE TO CLEARANCE
     Q365=0     ;TYPE OF TRAVERSE
```

LINEAR PATTERN (Cycle G221)

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

Sequence:

- Move to 2nd setup clearance (spindle)
- 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.

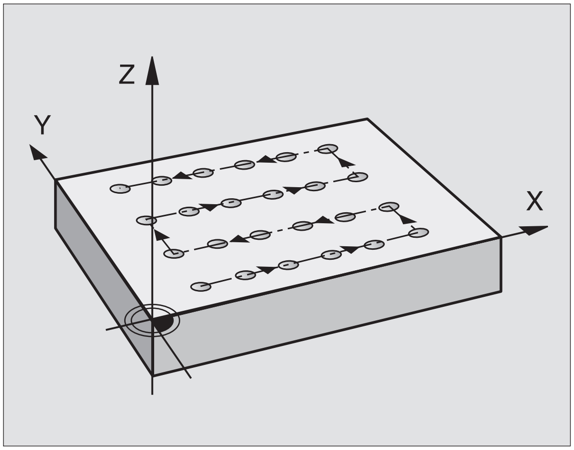
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.

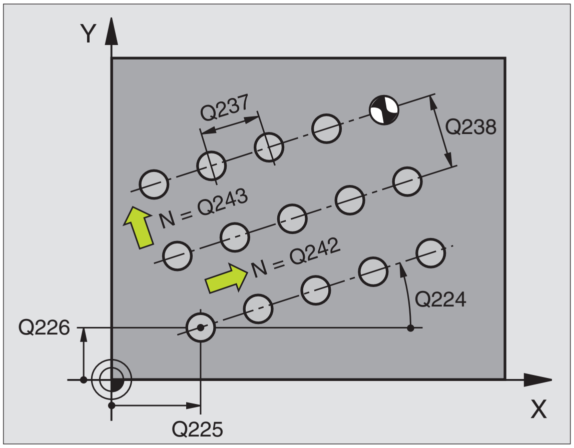
6 From this position the tool approaches the starting point for the next machining operation in the negative reference axis direction.

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.



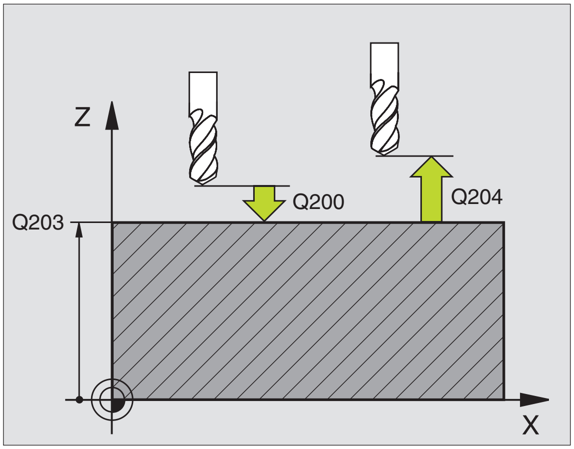


Before programming, note the following:

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.

The slot position 0 is not allowed if you use Cycle 254 Circular Slot in together with Cycle 221.

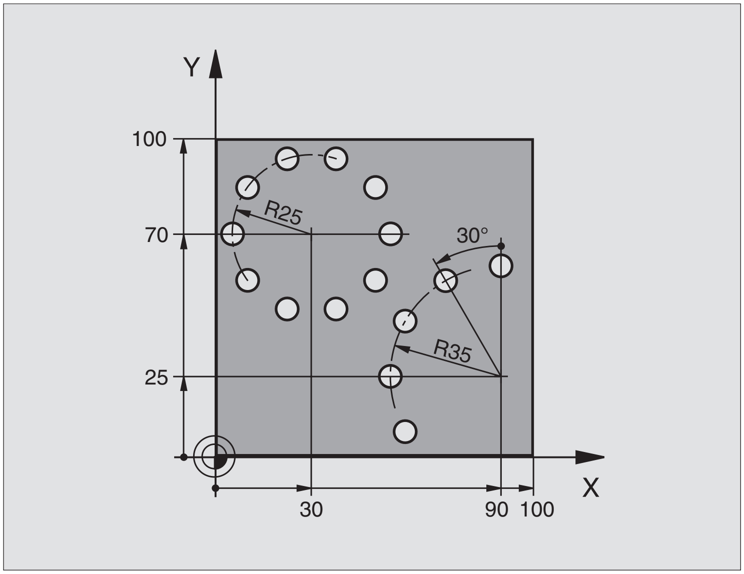


- **Starting point 1st axis** Q225 (absolute value): Coordinate of the starting point in the reference axis of the working plane.
- **Starting point 2nd axis** Q226 (absolute value): Coordinate of the starting point in the minor axis of the working plane.
- **Spacing in 1st axis** Q237 (incremental value): 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 machining operations.

Example: NC blocks

```
N540 G221 CARTESIAN PATTERN
     Q225=+15    ;STARTING POINT 1ST AXIS
     Q226=+15    ;STARTING POINT 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    ;ROTATIONAL POSITION
     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 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+70 ;CENTER 2ND AXIS	
Q244=50 ;PITCH CIRCLE DIA.	
Q245=+0 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+0 ;STEPPING ANGLE	
Q241=10 ;QUANTITY	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=1 ;TYPE OF TRAVERSE	
N80 G220 POLAR PATTERN	Define cycle for circular pattern 2, CYCL 200 is called automatically,
Q216=+90 ;CENTER 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+25 ;CENTER 2ND AXIS	
Q244=70 ;PITCH CIRCLE DIA.	
Q245=+90 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=30 ;STEPPING ANGLE	
Q241=5 ;QUANTITY	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=1 ;TYPE OF TRAVERSE	
N90 G00 G40 Z+250 M02 *	Retract in the tool axis, end program
N99999999 %PATTERN G71 *	

8.6 SL Cycles

Fundamentals

SL cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle **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. 8192 line blocks.
>
> SL cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always run a graphical program test before machining! This is a simple way of finding out whether the TNC-calculated program will provide the desired results.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation **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 in useful combinations. Always define both axes of the machining plane in the first block.
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms.

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 *
...
N99999999 %SL2 G71 *
```

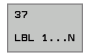
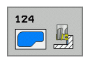
Characteristics of the fixed cycles

- 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.
- In order to avoid leaving dwell marks, the TNC inserts a globally definable rounding radius at non-tangential inside corners. The rounding radius, which is entered in Cycle G20, affects the tool center point path, meaning that it would increase a rounding defined by the tool radius (applies to rough-out and side finishing).
- 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.

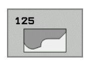
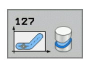
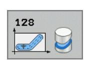
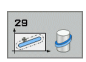
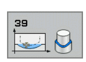
With MP7420 you can determine where the tool is positioned at the end of Cycles G121 to G124.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle **G120.**

Overview of SL cycles

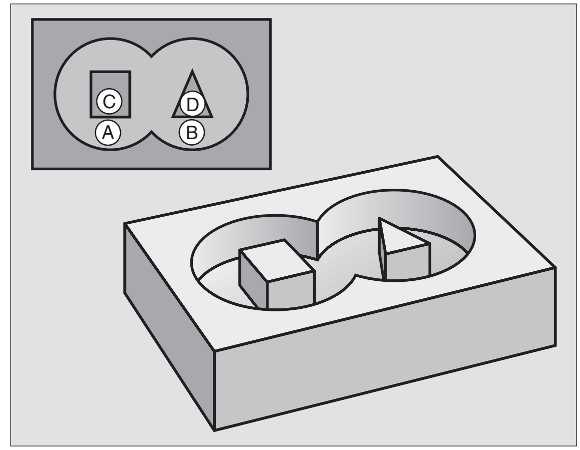
Cycle	Soft key	Page
G37 CONTOUR GEOMETRY (essential)		Page 386
G120 CONTOUR DATA (essential)		Page 390
G121 PILOT DRILLING (optional)		Page 391
G122 ROUGH-OUT (essential)		Page 392
G123 FLOOR FINISHING (optional)		Page 394
G124 SIDE FINISHING (optional)		Page 395

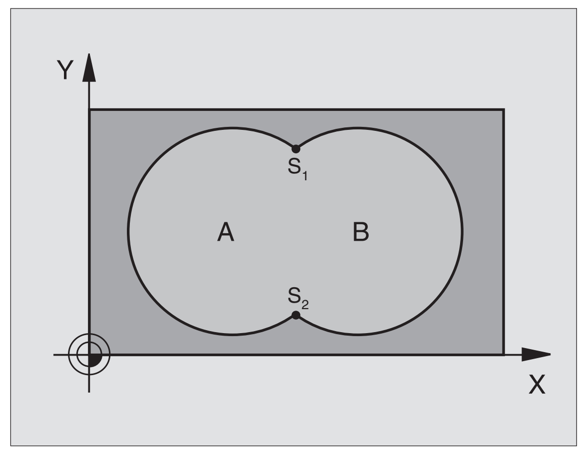
Enhanced cycles:

Cycle	Soft key	Page
G125 CONTOUR TRAIN		Page 396
G127 CYLINDER SURFACE		Page 398
G128 CYLINDER SURFACE slot milling		Page 400
G129 CYLINDER SURFACE ridge milling		Page 402
G139 CYLINDER SURFACE outside contour milling		Page 404

CONTOUR GEOMETRY (Cycle G37)

All subprograms that are superimposed to define the contour are listed in Cycle **G37** CONTOUR GEOMETRY.

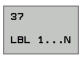




Before programming, note the following:

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 Y+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50 *
N550 G98 L0 *
```

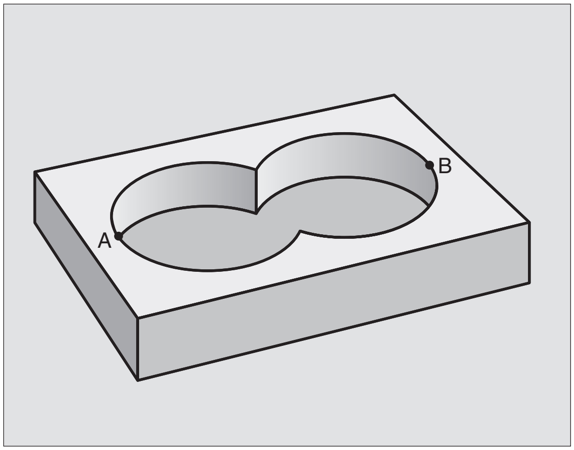
Subprogram 2: Pocket B

```
N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50 *
N590 G02 X+90 Y+50 *
N600 G90 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 *
```

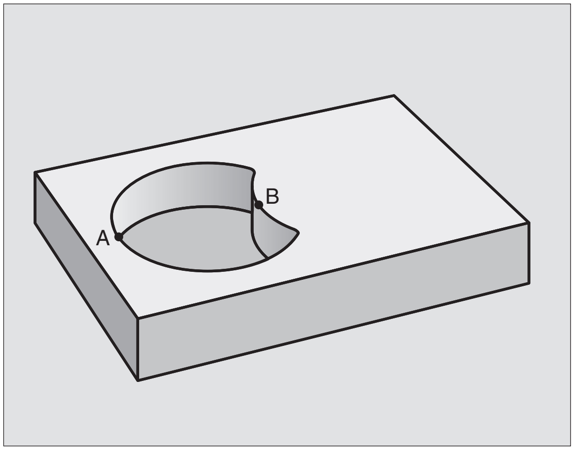
Surface B:

```
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 *
```

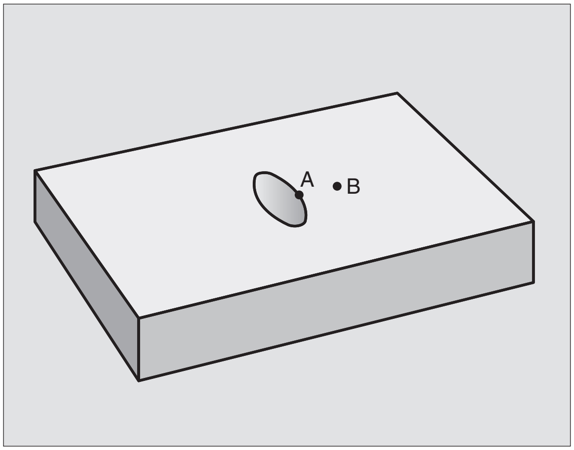
Surface B:

```
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.**

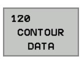
Before programming, note the following:

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.

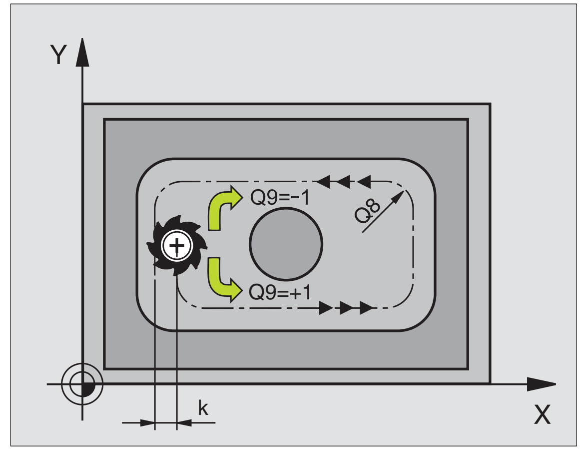
The machining data entered in Cycle **G120** are valid for Cycles G121 to G124.

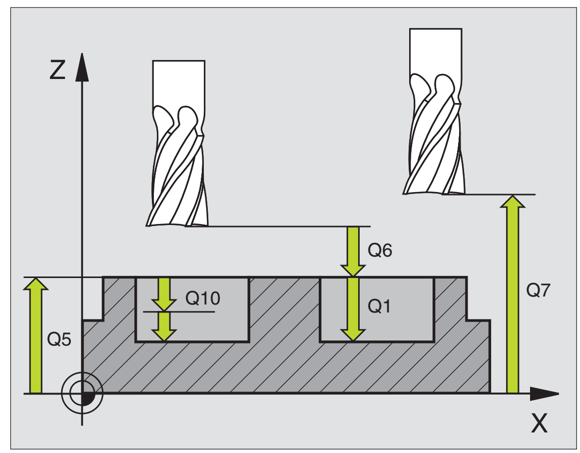
If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q19 cannot be used as program parameters.



- **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 HEIGHT
    Q8=0.5      ;ROUNDING RADIUS
    Q9=+1       ;DIRECTION
```

PILOT DRILLING (Cycle G121)

Process

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

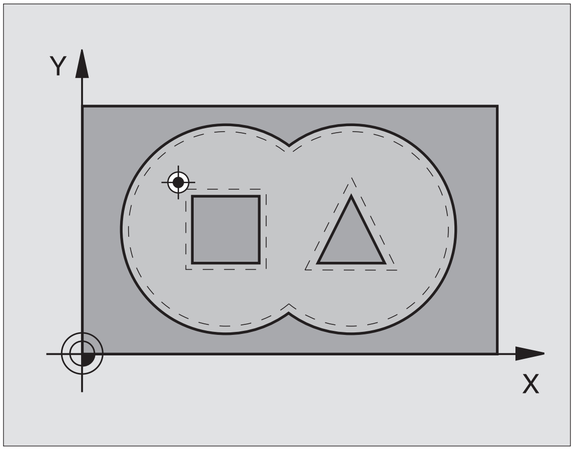
2 When it reaches the first plunging depth, the tool retracts at rapid traverse to the starting position and advances again to the first plunging depth minus the advanced stop distance t.

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.

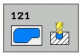
5 The TNC repeats this process (1 to 4) until the programmed depth is reached.

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



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.

Before programming, note the following:

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.

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)

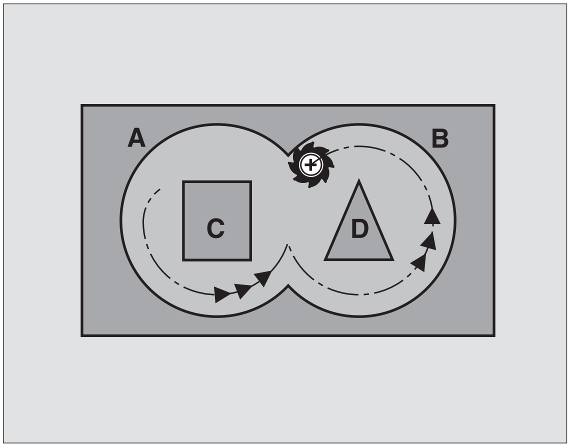
1 The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.

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 In the next step the TNC moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached.

5 Finally the TNC retracts the tool to the clearance height.



Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle `G121`.

You define the plunging behavior of Cycle 22 with parameter Q19 and with the tool table in the ANGLE and LCUTS columns:

- If Q19=0 is defined, the TNC always plunges perpendicularly, even if a plunge angle (ANGLE) is defined for the active tool.
- If you define the ANGLE=90°, the TNC plunges perpendicularly. The reciprocation feed rate Q19 is used as plunging feed rate.
- If the reciprocation feed rate Q19 is defined in Cycle 22 and ANGLE is defined between 0.1 and 89.999 in the tool table, the TNC plunges helically at the defined ANGLE.
- If the reciprocation feed is defined in Cycle 22 and no ANGLE is in the tool table, the TNC displays an error message.
- If geometrical conditions do not allow helical plunging (slot geometry), the TNC tries a reciprocating plunge. The reciprocation length is calculated from LCUTS and ANGLE (reciprocation length = LCUTS / tan ANGLE)

If you clear out an acute inside corner and use an overlap factor greater than 1, some material might be left over. Check especially the innermost path in the test run graphic and, if necessary, change the overlap factor slightly. This allows another distribution of cuts, which often provides the desired results.

- **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** Q18 or QS18: Number or name of the tool with which the TNC has already coarse-roughed the contour. Switch to name input: Press the **"** key. If there was no coarse roughing, enter "0"; if you enter a number or a name, 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 181 and define the maximum plunging ANGLE of the tool. The TNC will otherwise generate an error message.
- **Reciprocation feed rate** Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut.
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting after machining. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q12.
- **Feed rate factor in %:** Q401: Percentage factor by which the TNC reduces the machining feed rate **(Q12)** as soon as the tool moves within the material over its entire circumference during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap(**Q2**) specified in Cycle 20. The TNC then reduces the feed rate as per your definition at transitions and narrow places, so the machine time should be reduced in total.

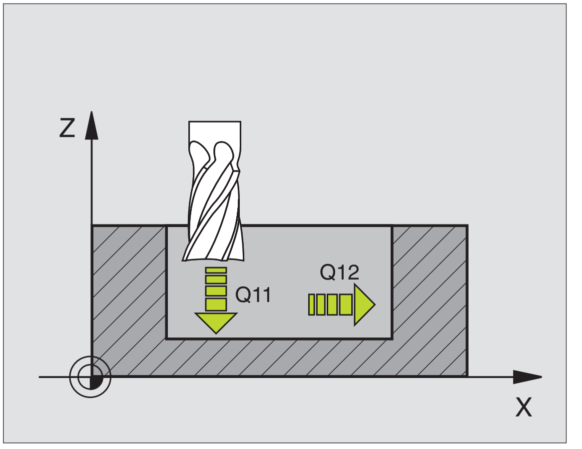
> Feed rate reduction through parameter Q401 is an FCL3 function and is not automatically available after a software update (see "Feature content level (upgrade functions)" on page 8).

Example: NC block

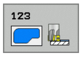
```
N59 G122 ROUGH-OUT
    Q10=+5      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLUNGING
    Q12=350     ;FEED RATE FOR ROUGHING
    Q18=1       ;COARSE ROUGHING TOOL
    Q19=150     ;RECIPROCATION FEED RATE
    Q208=99999  ;RETRACTION FEED RATE
    Q401=80     ;FEED RATE REDUCTION
```

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.
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting after machining. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q12.

Example: NC block

```
N60 G123 FLOOR FINISHING
    Q11=100    ;FEED RATE FOR PLUNGING
    Q12=350    ;FEED RATE FOR ROUGHING
    Q208=99999 ;RETRACTION FEED RATE
```

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:

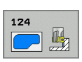
The sum of allowance for side (Q14) and the radius of the finish mill must be smaller than the sum of allowance for side (Q3, Cycle **G120**) and the radius of the rough mill.

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.

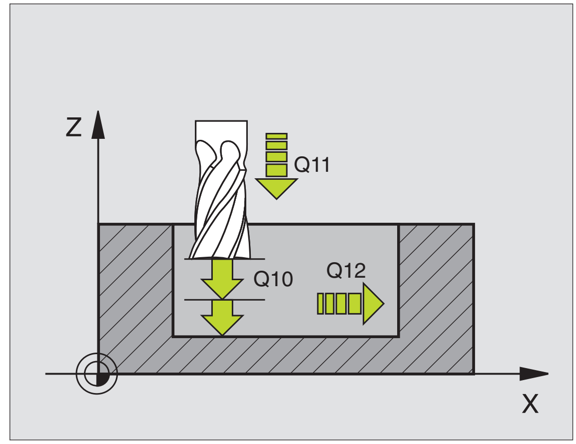
You can use Cycle **G124** also for contour milling. Then you must:

- define the contour to be milled as a single island (without pocket limit), and
- enter the finishing allowance (Q3) in Cycle G120 to be greater than the sum of the finishing allowance Q14 + radius of the tool being used.

The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket and the allowance programmed in Cycle G120.



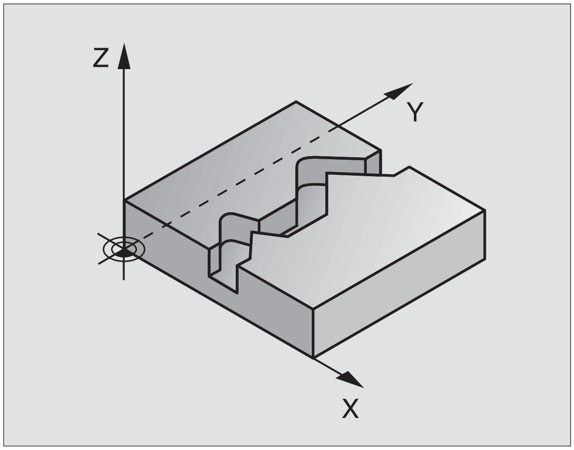
- **Direction of rotation ? Clockwise = -1** Q9: Machining direction:
  +1: Counterclockwise
  -1: Clockwise
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration.
- **Feed rate 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
    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.

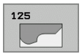
Cycle **G120** CONTOUR DATA is not required.

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.



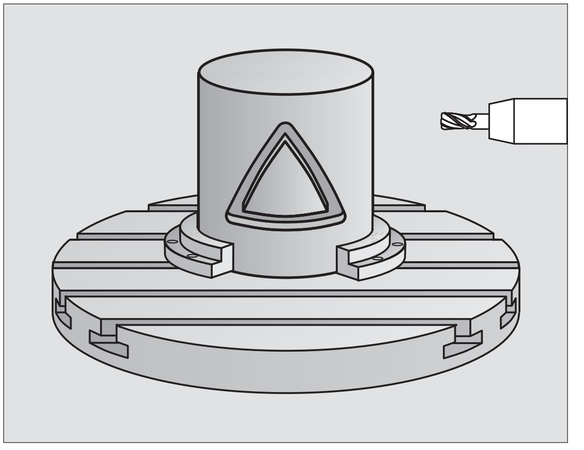
- **Milling depth** Q1 (incremental value): Distance between workpiece surface and contour floor.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the working plane.
- **Workpiece surface coordinate** Q5 (absolute value): Absolute coordinate of the workpiece surface referenced 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 HEIGHT
    Q10=+5      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLUNGING
    Q12=350     ;FEED RATE FOR MILLNG
    Q15=-1      ;CLIMB OR UP-CUT
```

CYLINDER SURFACE (Cycle G127, software option 1)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

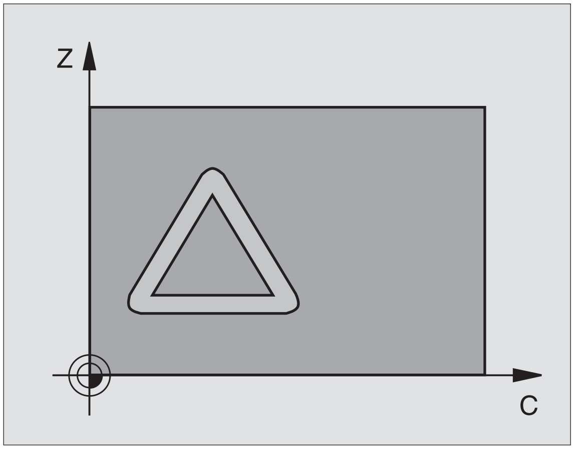


This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle **G128** if you wish to mill guide notches onto the cylinder surface.

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. You can program up to 8192 contour elements 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.

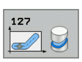
This cycle requires a center-cut end mill (ISO 1641).

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.



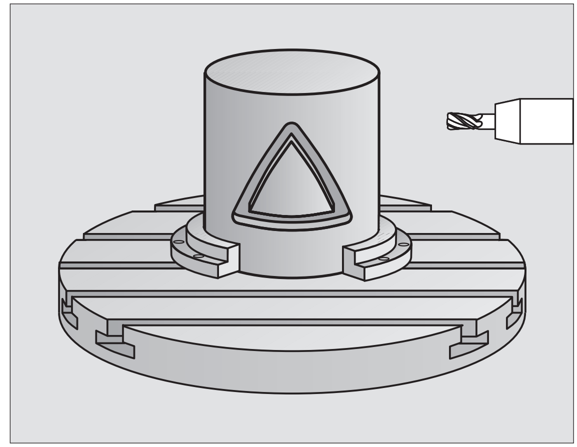
- **Milling depth** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius 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 MILLNG
    Q16=25        ;RADIUS
    Q17=0         ;TYPE OF DIMENSION
```

CYLINDER SURFACE slot milling (Cycle G128, software option 1)

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:

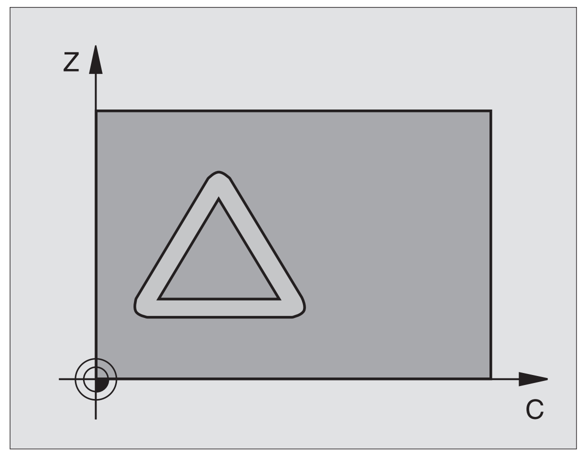
1 The TNC positions the tool over the cutter infeed point.

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:

In the first NC block of the contour program, always program both cylinder surface coordinates.

The memory capacity for programming an SL cycle is limited. You can program up to 8192 contour elements 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.

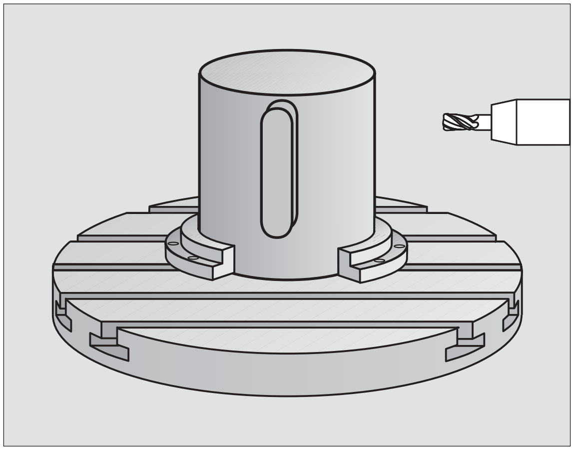
- **Milling depth** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius 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.
- **Tolerance?** Q21: If you use a tool smaller than the programmed slot width Q20, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance Q21, the TNC adds a subsequent milling operation to ensure that the slot dimensions are a close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With Q21 you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the remachining takes. **Recommendation:** Use a tolerance of 0.02 mm.
  0: Function inactive

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 MILLNG
    Q16=25       ;RADIUS
    Q17=0        ;TYPE OF DIMENSION
    Q20=12       ;SLOT WIDTH
    Q21=0        ;TOLERANCE
```

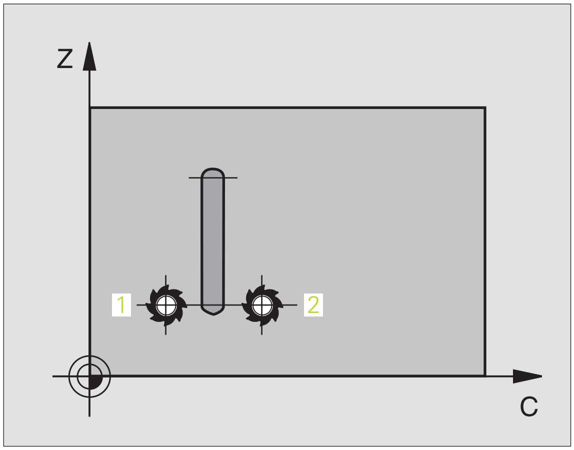
CYLINDER SURFACE ridge milling (Cycle G129, software option 1)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.



This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. 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 ridge together with the tool radius compensation. With the radius compensation you specify whether the TNC cuts the ridge with climb milling or up-cut milling.

At the ends of the ridge the TNC always adds a semicircle whose radius is half the ridge width.



1 The TNC positions the tool over the starting point of machining. The TNC calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins from the left (**1**, RL = climb milling) or the right of the ridge (**2**, RR = up-cut milling) (see figure at center right).

2 After the TNC has positioned to the first plunging depth, the tool moves on a circular arc at the milling feed rate Q12 tangentially to the ridge wall. If so programmed, it will leave metal for the finishing allowance.

3 At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate Q12 until the stud is completed.

4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.

5 Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.

6 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on machine parameter 7420).

Before programming, note the following:

In the first NC block of the contour program, always program both cylinder surface coordinates.

Ensure that the tool has enough space laterally for contour approach and departure.

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.

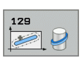
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

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.



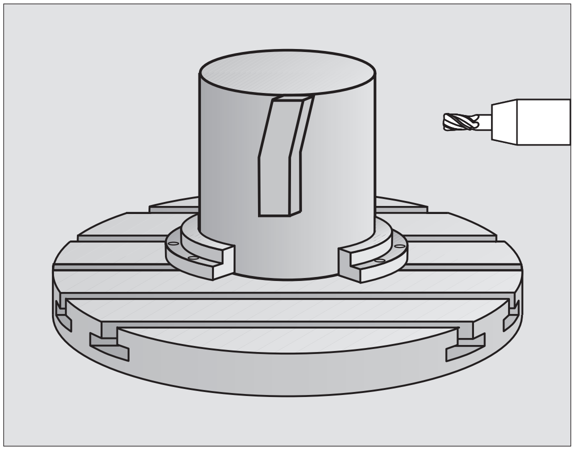
- **Milling depth** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value.
- **Set-up 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).
- **Ridge width** Q20: Width of the ridge to be machined.

Example: NC blocks

```
N50 G129 CYLINDER SURFACE RIDGE
    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 MILLNG
    Q16=25       ;RADIUS
    Q17=0        ;TYPE OF DIMENSION
    Q20=12       ;RIDGE WIDTH
```

CYLINDER SURFACE outside contour milling (Cycle G139, software option 1)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.



This cycle enables you to program an open contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. With this cycle the TNC adjusts the tool so that, with radius compensation active, the wall of the open contour is always parallel to the cylinder axis.

Unlike Cycles 28 and 29, in the contour subprogram you define the actual contour to be machined.

1. The TNC positions the tool over the starting point of machining. The TNC locates the starting point next to the first point defined in the contour subprogram, offset by the tool diameter.
2. After the TNC has positioned to the first plunging depth, the tool moves on a circular arc at the milling feed rate Q12 tangentially to the contour. If so programmed, it will leave metal for the finishing allowance.
3. At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12 until the contour train is completed.
4. The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5. Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.
6. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle (depending on machine parameter 7420).

Before programming, note the following:

Ensure that the tool has enough space laterally for contour approach and departure.

The memory capacity for programming an SL cycle is limited. You can program up to 8192 contour elements in one SL cycle.

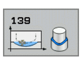
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

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.

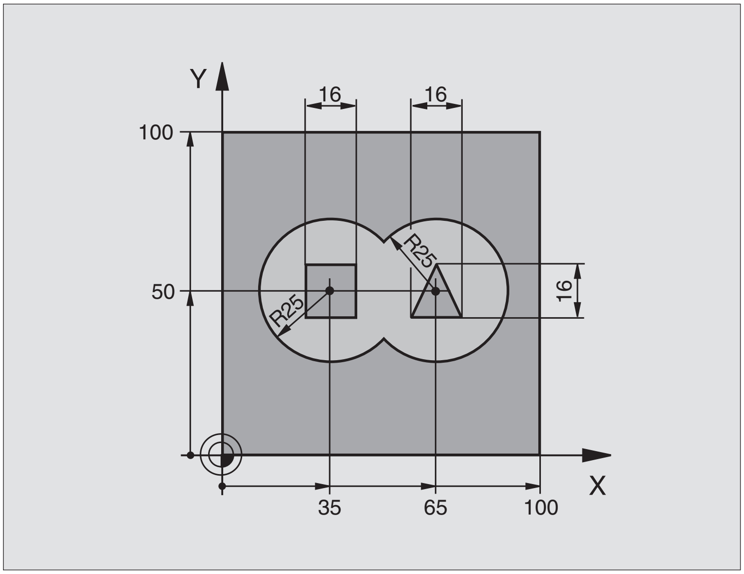


- **Milling depth** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance on the contour wall.
- **Set-up clearance** Q6 (incremental value): Distance between the tool tip and the cylinder 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 blocks

```
N50 G139 CYL. SURFACE CONTOUR
    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 MILLNG
    Q16=25       ;RADIUS
    Q17=0        ;TYPE OF DIMENSION
```

Example: Pilot drilling, roughing-out and finishing overlapping contours

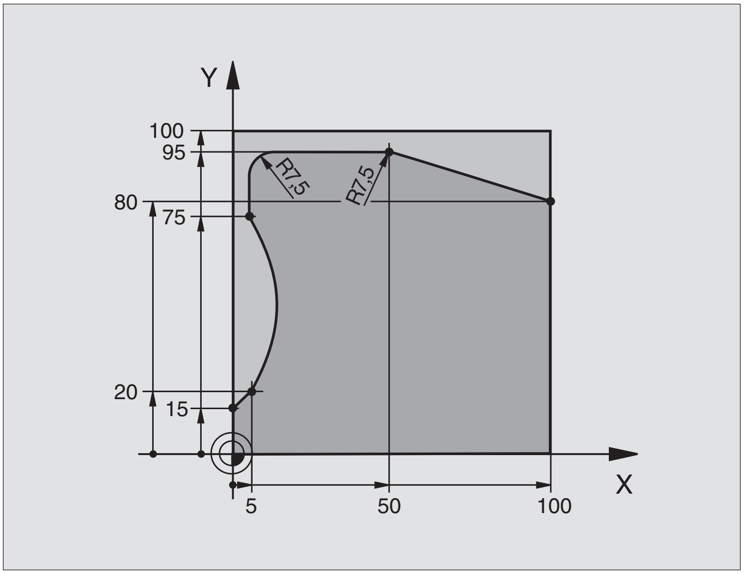


%C21 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+6 *	Define tool: drill
N40 G99 T2 L+0 R+6 *	Define the tool for roughing/finishing
N50 T1 G17 S4000 *	Call tool: drill
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 P02 2 P03 3 P04 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 HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION	

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	
Q208=2000 ;RETRACTION 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	
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+25 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 L0 *	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 *	
N99999999 %C21 G71 *	

Example: Contour train



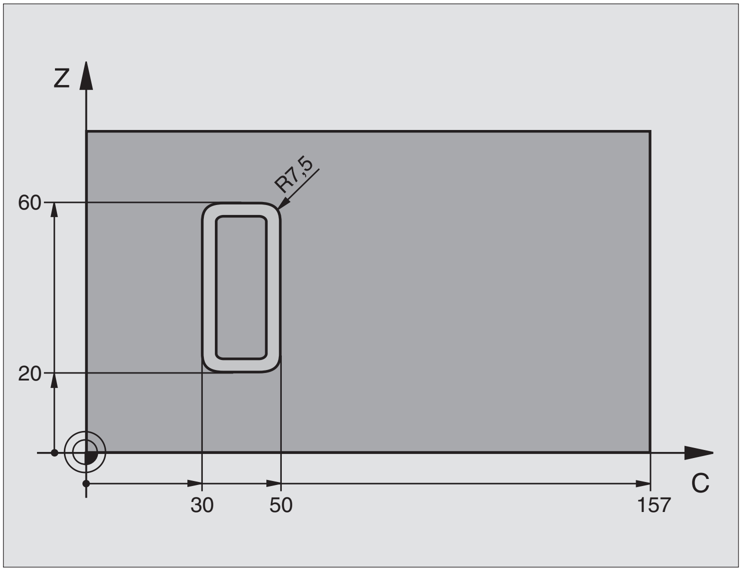
%C25 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define the tool
N40 T1 G17 S2000 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G37 P01 1 *	Define contour subprogram
N70 G125 CONTOUR TRAIN	Define machining parameters
Q1=-20 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q5=+0 ;SURFACE COORDINATE	
Q7=+250 ;CLEARANCE HEIGHT	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLUNGING	
Q12=200 ;FEED RATE FOR MILLNG	
Q15=+1 ;CLIMB OR UP-CUT	
N80 G79 M3 *	Call the cycle
N90 G00 G90 Z+250 M2 *	Retract in the tool axis, end program

`N100 G98 L1 *`	Contour subprogram
`N110 G01 G41 X+0 Y+15 *`	
`N120 X+5 Y+20 *`	
`N130 G06 X+5 Y+75 *`	
`N140 G01 Y+95 *`	
`N150 G25 R7.5 *`	
`N160 X+50 *`	
`N170 G25 R7.5 *`	
`N180 X+100 Y+80 *`	
`N190 G98 L0 *`	
`N99999999 %C25 G71 *`	

Example: Cylinder surface with Cycle G127

Note:

- Cylinder centered on rotary table
- Datum at center of rotary table



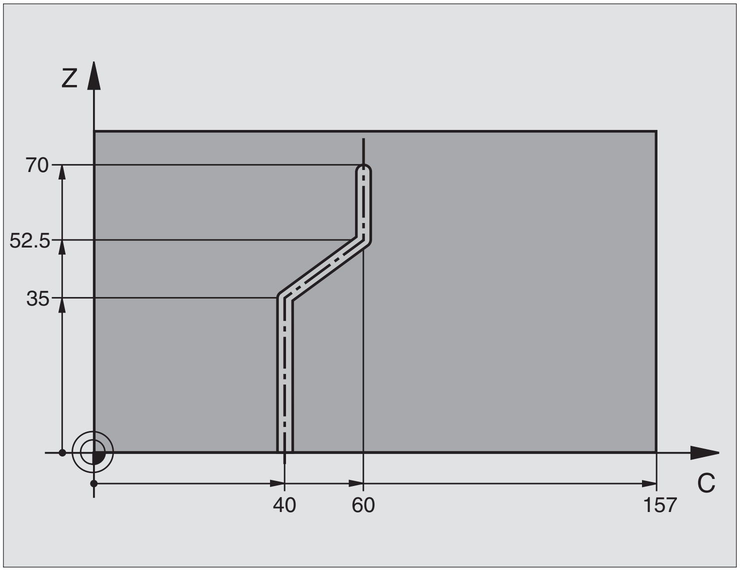
`%C27 G71 *`	
`N10 G99 T1 L+0 R3.5 *`	Define the tool
`N20 T1 G18 S2000 *`	Call tool, tool axis is Y
`N30 G00 G40 G90 Y+250 *`	Retract the tool
`N40 G37 P01 1 *`	Define contour subprogram
`N70 G127 CYLINDER SURFACE`	Define machining parameters
`Q1=-7 ;MILLING DEPTH`	
`Q3=+0 ;ALLOWANCE FOR SIDE`	
`Q6=2 ;SET-UP CLEARANCE`	
`Q10=4 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=250 ;FEED RATE FOR MILLNG`	
`Q16=25 ;RADIUS`	
`Q17=1 ;TYPE OF DIMENSION`	
`N60 C+0 M3 *`	Pre-position rotary table
`N70 G79 *`	Call the cycle
`N80 G00 G90 Z+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 *	
N99999999 %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 R3.5 *`	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 MILLNG`	
`Q16=25 ;RADIUS`	
`Q17=1 ;TYPE OF DIMENSION`	
`Q20=10 ;SLOT WIDTH`	
`Q21=0.02 ;TOLERANCE`	
`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
`N110 G01 G41 C+40 Z+0 *`	Data for the rotary axis are entered in mm (Q17=1)
`N120 Z+35 *`	
`N130 C+60 Z+52.5 *`	
`N140 Z+70 *`	
`N150 G98 L0 *`	
`N99999999 %C28 G71 *`	

8.7 SL Cycles with Contour Formulas

Fundamentals

SL cycles and the contour formulas 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 **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.

The SL cycles with contour formulas presuppose a structured program layout and enable you to save frequently used contours in individual programs. Using a 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 formulas” 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 formulas

```
%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 formulas

```
%MODEL G71 *
N10 DECLARE CONTOUR QC1 = "CIRCLE1" *
N20 DECLARE CONTOUR QC2 = "CIRCLE31XY" *
N30 DECLARE CONTOUR QC3 = "TRIANGLE" *
N40 DECLARE CONTOUR QC4 = "SQUARE" *
N50 QC10 = ( QC1 | QC3 | QC4 ) \ QC2 *
N99999999 %MODEL G71 *

%CIRCLE1 G71 *
N10 I+75 J+50 *
N20 G11 R+45 H+0 G40 *
N30 G13 G91 H+360 *
N99999999 %CIRCLE1 G71 *

%CIRCLE31XY 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 set-up 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

SELECT CONTOUR

- ▶ To select the functions for program call, press the PGM CALL key.
- ▶ 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

CONTOUR

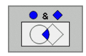
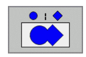
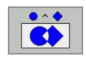
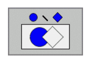
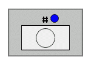
- ▶ Press the DECLARE soft key.
- ▶ Press the CONTOUR soft key.
- ▶ Enter the number for the contour designator **QC,** and confirm with the ENT key.
- ▶ Enter the full name of the program with the contour description and confirm with the END key.

> 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)`**	)
Defining a single contour e.g. **`QC12 = QC1`**	

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 *
N99999999 %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 *
N99999999 %POCKET_B G71 *
```

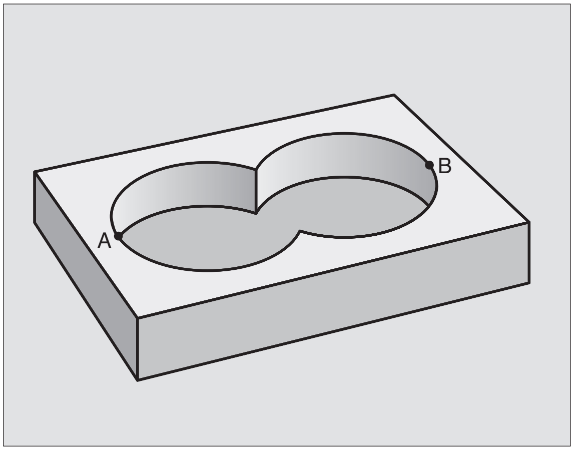
Area of inclusion

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

- The surfaces A and B must be entered 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 ...
```



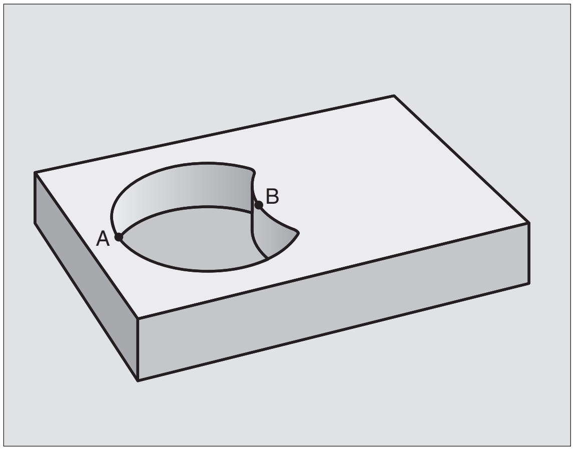
Area of exclusion

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

- The surfaces A and B must be entered 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 ...
```



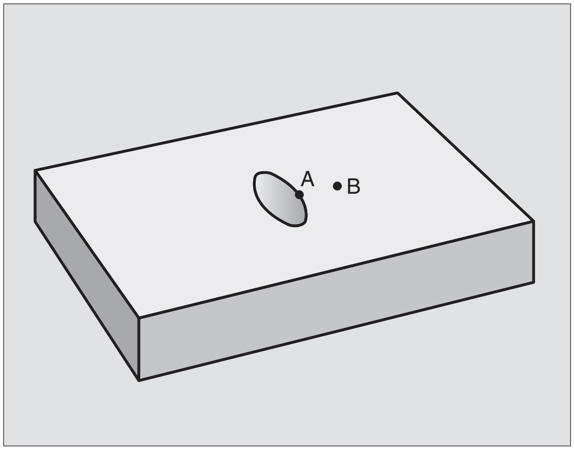
Area of intersection

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 entered 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:

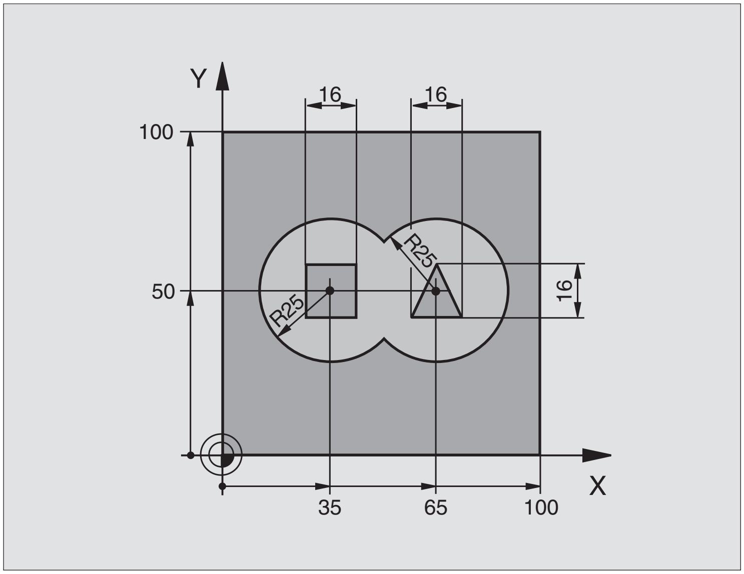
```
N50 ...
N60 ...
N70 DECLARE CONTOUR QC1 = "POCKET_A.H" *
N80 DECLARE CONTOUR QC2 = "POCKET_B.H" *
N90 QC10 = QC1 & QC2 *
N100 ...
N110 ...
```



Contour machining with SL Cycles

The complete contour is machined with the SL Cycles G120 to G124 (see "SL Cycles" on page 383).

Example: Roughing and finishing superimposed contours with a 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 HEIGHT`	
`Q8=0.1 ;ROUNDING RADIUS`	
`Q9=-1 ;DIRECTION`	

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	
Q208=750 ;RETRACTION 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	
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
N99999999 %C21 G71 *	

Contour definition program with contour formula:

%MODEL G71 *	Contour definition program
N10 DECLARE CONTOUR QC1 = "CIRCLE1" *	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 = "CIRCLE31XY" *	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 QC1 = "SQUARE" *	Definition of the contour designator for the program “SQUARE”
N80 QC10 = (QC1 | QC2) \ QC3 \ QC4 *	Contour formula
N99999999 %MODEL G71 *	

Contour description programs:

`%CIRCLE1 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 %CIRCLE1 G71 *`	

`%KREOS31XY G71 *`	Contour description program: circle at left
`N10 I+Q1 J+Q2 *`	
`N20 G11 R+Q3 H+0 G40 *`	
`N30 G13 G91 H+360 *`	
`N99999999 %CIRCLE31XY 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 *`	
`N49 G01 X+73 *`	
`N99999999 %TRIANGLE G71 *`	

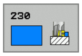
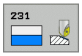
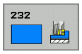
`%SQUARE G71 *`	Contour description program: square at left
`N10 G01 X+27 Y+58 G40 *`	
`N20 G01 X+43 *`	
`N30 G01 Y+42 *`	
`N40 G01 X+27 *`	
`N50 G01 Y+58 *`	
`N99999999 %SQUARE G71 *`	

8.8 Cycles for Multipass Milling

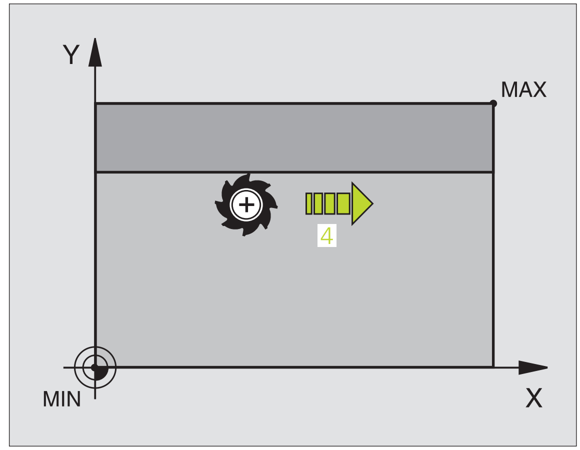
Overview

The TNC offers four cycles for machining surfaces with the following characteristics:

- 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	Page
G60 RUN 3-D DATA For multipass milling of 3-D data in several infeeds		Page 425
G230 MULTIPASS MILLING For flat rectangular surfaces		Page 426
G231 RULED SURFACE For oblique, inclined or twisted surfaces		Page 428
G232 FACE MILLING For level rectangular surfaces, with indicated oversizes and multiple infeeds		Page 431

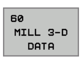
RUN 3-D DATA (Cycle G60)



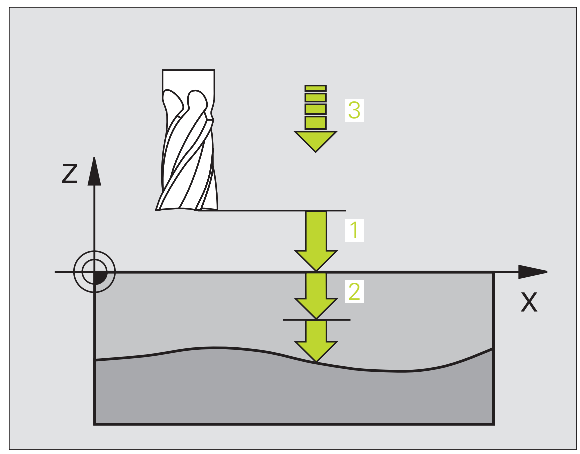
1 From the current position, the TNC positions the tool in rapid traverse in the tool axis to the set-up clearance above the MAX point that you have programmed in the cycle.
2 The tool then moves in rapid traverse in the working plane to the MIN point you have programmed in the cycle.
3 From this point, 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:

You can use Cycle 30 to run conversational programs created offline in multiple infeeds.



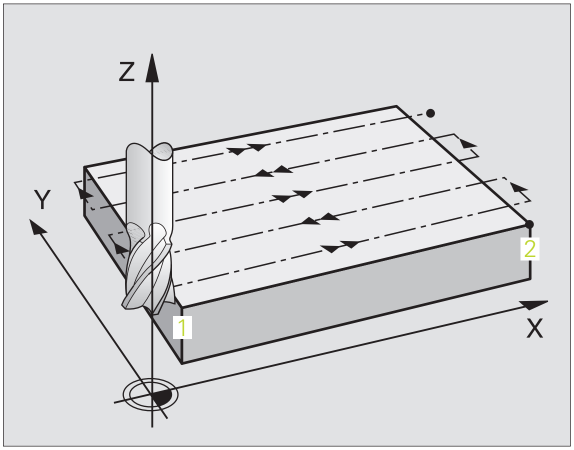
- **PGM Name 3-D data**: Enter the name of the file in which the data is stored. If the file is not stored in the current directory, enter the complete path.
- **Min. point of range:** Lowest coordinates (X, Y and Z coordinates) in the range to be milled.
- **Max. point of range:** Largest coordinates (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 *
```

MULTIPASS MILLING (Cycle G230)

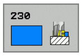


1. From the current position in the working plane, the TNC positions the tool at rapid traverse to the starting point **1**; the TNC moves the tool by its radius to the left and upward.
2. The tool then moves in rapid traverse in the tool axis to set-up clearance. From there it approaches the programmed 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.

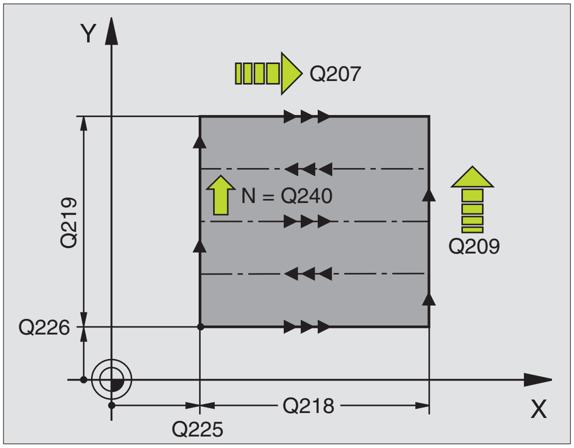
Before programming, note the following:

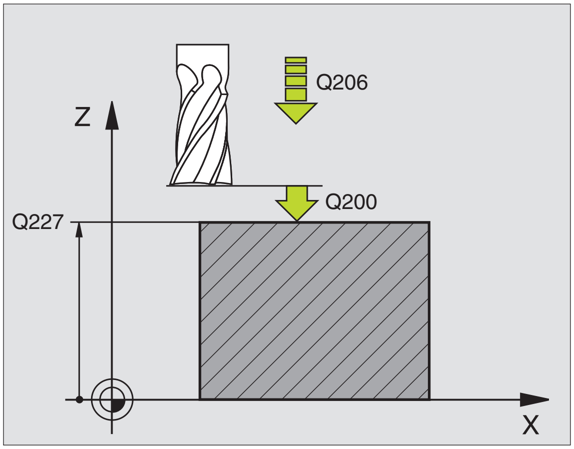
From the current position, the TNC positions the tool at the starting point, first in the working plane and then in the spindle axis.

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** Q206: Traversing speed of the tool in mm/min when moving from set-up clearance to the milling depth.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Stepover feed rate** Q209: Traversing speed of the tool in mm/min when moving to the next pass. If you are moving the tool transversely in the material, enter Q209 to be smaller than Q207. If you are moving it transversely in the open, Q209 may be greater than Q207.
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and milling depth for positioning at the start and end of the cycle.

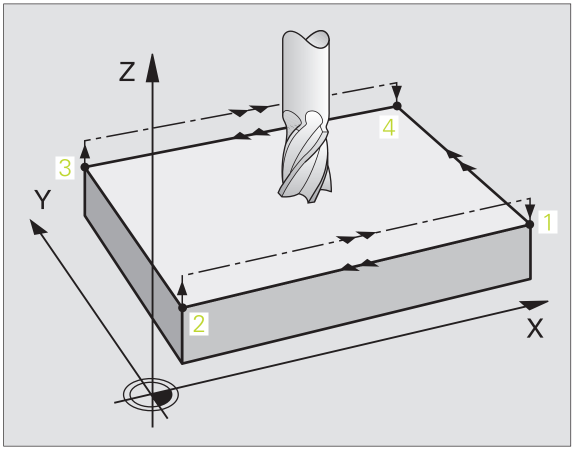




Example: NC block

```
N71 G230 MULTIPASS MILLING
    Q225=+10     ;STARTING POINT 1ST AXIS
    Q226=+12     ;STARTING POINT 2ND AXIS
    Q227=+2.5    ;STARTING POINT 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 MILLNG
    Q209=200     ;STEPOVER FEED RATE
    Q200=2       ;SET-UP CLEARANCE
```

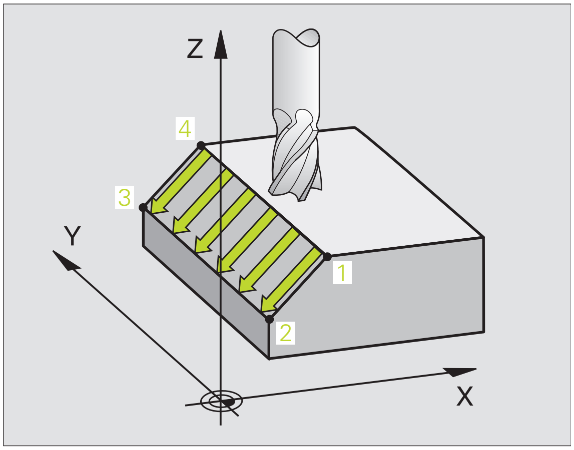
RULED SURFACE (Cycle G231)



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.

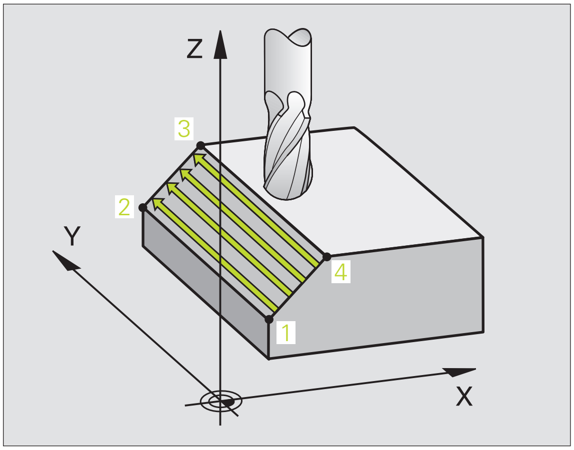


If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

- 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.

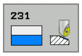


Before programming, note the following:

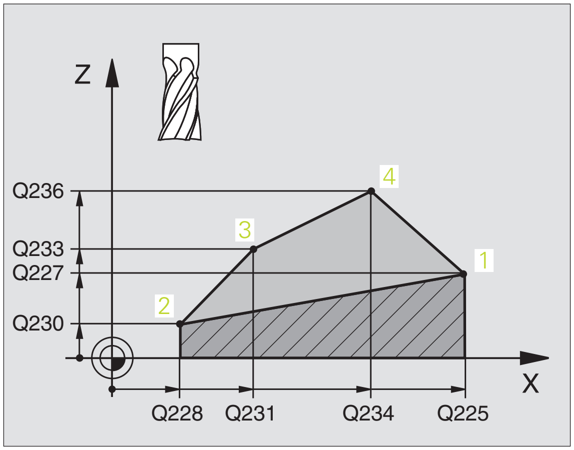
From the current position, the TNC positions the tool in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

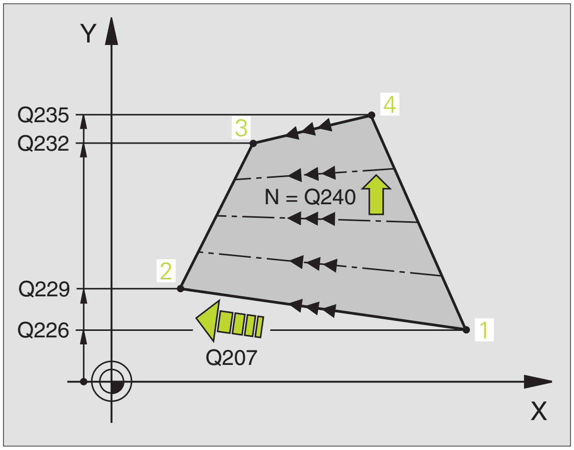
The TNC moves the tool with radius compensation **G40** to the programmed positions.

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 POINT 1ST AXIS
     Q226=+5     ;STARTING POINT 2ND AXIS
     Q227=-2     ;STARTING POINT 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 MILLNG
```

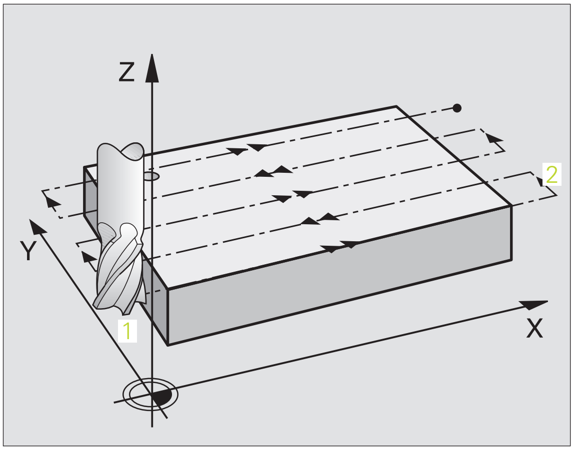
FACE MILLING (Cycle 232)

Cycle G232 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1:** Meander machining, stepover within the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

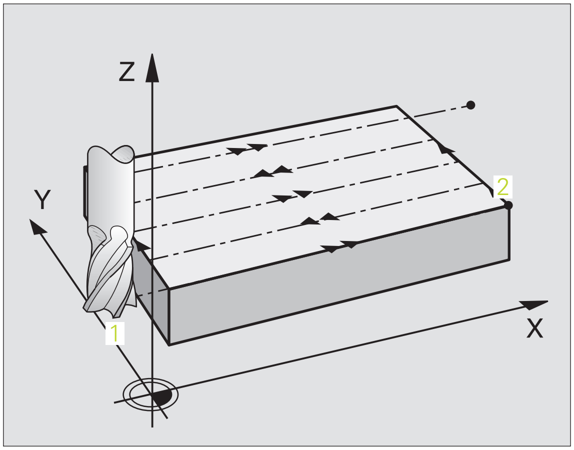
1 From the current position, the TNC positions the tool at rapid traverse to the starting position using positioning logic **1**: If the current position in the spindle axis is greater than the 2nd set-up clearance, the control positions the tool first in the machining plane and then in the spindle axis. Otherwise it first moves to the 2nd set-up clearance and then in the machining plane. The starting point in the machining plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.
2 The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

Strategy Q389=0



3 The tool then advances to the stopping point **2** at the feed rate for milling. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.
4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the next machining depth is plunged to.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at FMAX to the 2nd set-up clearance.

Strategy Q389=1



3 The tool then advances to the stopping point **2** at the feed rate for milling. The end point lies **within** the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.

4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.

5 The tool then moves back in the direction of the starting point **1**. The motion to the next line occurs within the workpiece borders.

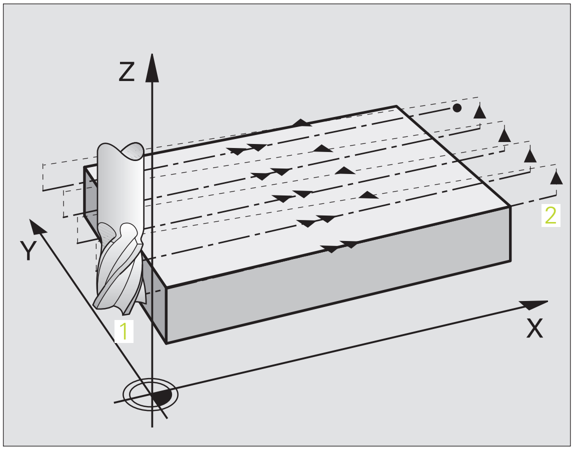
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the next machining depth is plunged to.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at FMAX to the 2nd set-up clearance.

Strategy Q389=2



3 The tool then advances to the stopping point **2** at the feed rate for milling. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.

4 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at the pre-positioning feed rate directly back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.

5 The tool then returns to the current infeed depth and moves in the direction of the next end point **2**

6 The milling process is repeated until the programmed surface has been completed. At the end of the last pass, the next machining depth is plunged to.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

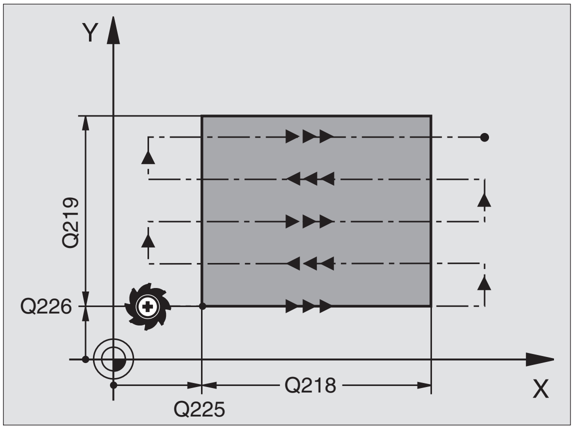
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

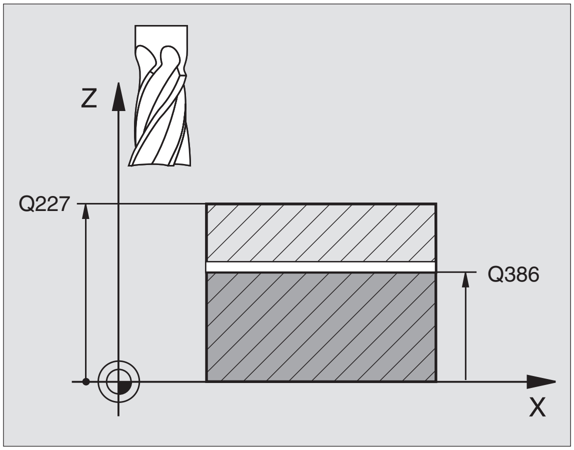
9 At the end of the cycle, the tool is retracted at FMAX to the 2nd set-up clearance.

Before programming, note the following:

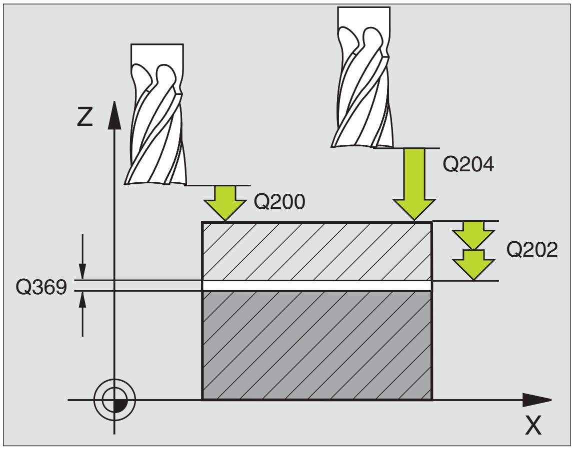
Enter the 2nd set-up clearance in Q204 so that no collision between tool and clamping devices can occur.

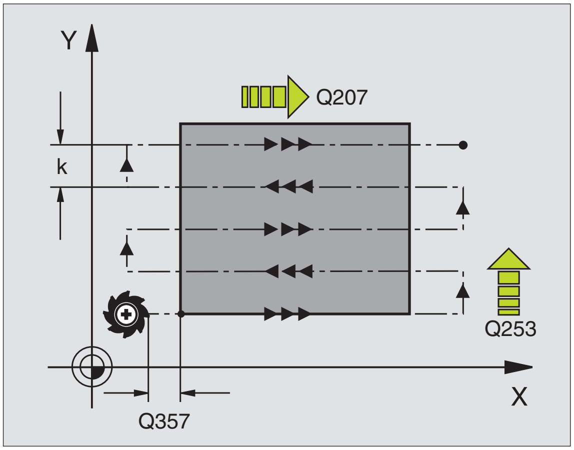
- **Machining strategy (0/1/2)** Q389: Specify how the TNC is to machine the surface:
  0: Meander machining, stepover at positioning feed rate outside the surface to be machined
  1: Meander machining, stepover at feed rate for milling within the surface to be machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate
- **Starting point in 1st axis** Q225 (absolute value): Starting point coordinate of the surface to be machined in the reference axis of the working plane.
- **Starting point in 2nd axis** Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane.
- **Starting point in 3rd axis** Q227 (absolute value): Coordinate of the workpiece surface used to calculate the infeeds.
- **End point in 3rd axis** Q386 (absolute value): Coordinate in the spindle axis to which the surface is to be face milled.
- **First side length** Q218 (incremental value): Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path in reference to the **starting point in the 1st axis.**
- **Second side length** Q219 (incremental value): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **starting point in the 2nd axis.**





- **Maximum plunging depth** Q202 (incremental value): **Maximum** amount that the tool is advanced each time. The TNC calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time.
- **Allowance for floor** Q369 (incremental value): Distance used for the last infeed.
- **Max. path overlap factor** Q370: **Maximum** stepover factor k. The TNC calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. tooth radius when using a face-milling cutter), the TNC reduces the stepover accordingly.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min while milling the last infeed.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely to the material (Q389=1), the TNC moves the tool at the feed rate for milling Q207.



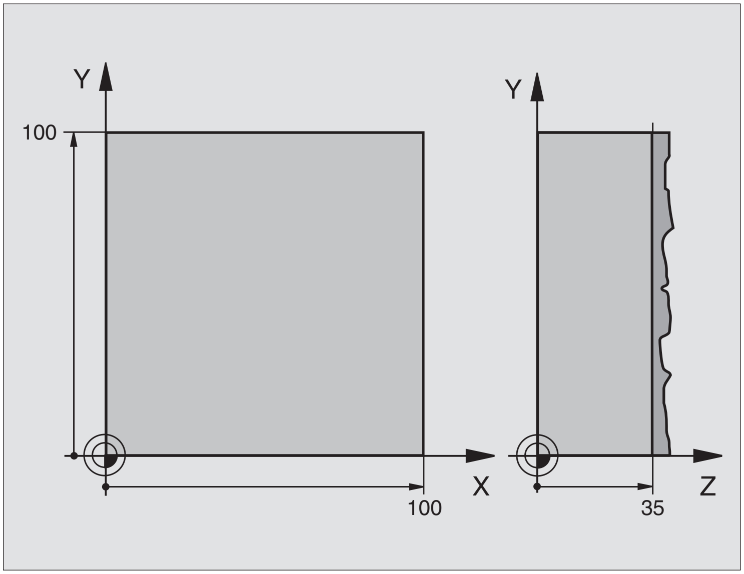


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy Q389=2, the TNC moves the tool at the set-up clearance over the current plunging depth to the starting point of the next pass.
- **Clearance to side** Q357 (incremental value): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

Example: NC blocks

```
N70 G232 FACE MILLING
    Q389=2      ;STRATEGY
    Q225=+10    ;STARTING POINT 1ST AXIS
    Q226=+12    ;STARTING POINT 2ND AXIS
    Q227=+2.5   ;STARTING POINT 3RD AXIS
    Q386=-3     ;END POINT IN 3RD AXIS
    Q218=150    ;FIRST SIDE LENGTH
    Q219=75     ;SECOND SIDE LENGTH
    Q202=2      ;MAX. PLUNGING DEPTH
    Q369=0.5    ;ALLOWANCE FOR FLOOR
    Q370=1      ;MAX. OVERLAP
    Q207=500    ;FEED RATE FOR MILLNG
    Q385=800    ;FEED RATE FOR FINISHING
    Q253=2000   ;F PRE-POSITIONING
    Q200=2      ;SET-UP CLEARANCE
    Q357=2      ;CLEARANCE TO SIDE
    Q204=2      ;2ND SET-UP CLEARANCE
```

Example: Multipass milling

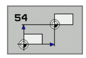
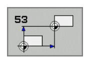
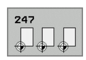
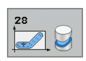
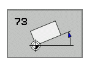
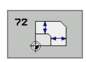


%C230 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+5 *	Define the tool
N40 T1 G17 S3500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G230 MULTIPASS MILLING	Cycle definition: MULTIPASS MILLING
Q225=+0 ;STARTNG PNT 1ST AXIS	
Q226=+0 ;STARTNG PNT 2ND AXIS	
Q227=+35 ;STARTNG 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
N99999999 %C230 G71 *	

8.9 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	Page
G54 DATUM For shifting contours directly within the program		Page 439
G53 DATUM from datum table		Page 440
G247 DATUM SETTING Datum setting during program run		Page 443
G28 MIRROR IMAGE Mirroring contours		Page 444
G73 ROTATION For rotating contours in the working plane		Page 446
G72 SCALING FACTOR For increasing or reducing the size of contours		Page 447
G80 WORKING PLANE Machining in tilted coordinate system on machines with swivel heads and/or rotary tables		Page 448

Effect of coordinate transformations

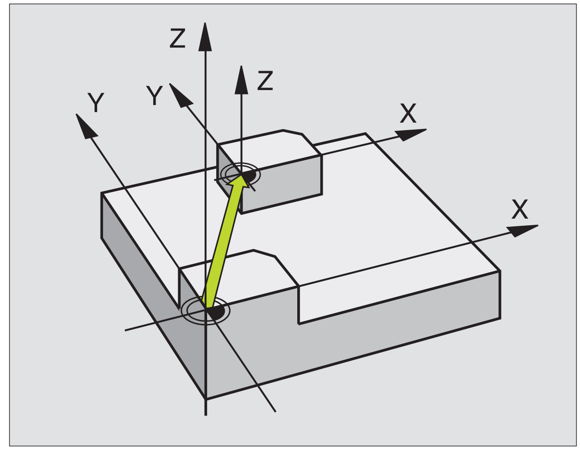
Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called. It remains in effect until it is changed or canceled.

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.

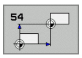
DATUM SHIFT (Cycle G54)

A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.



Effect

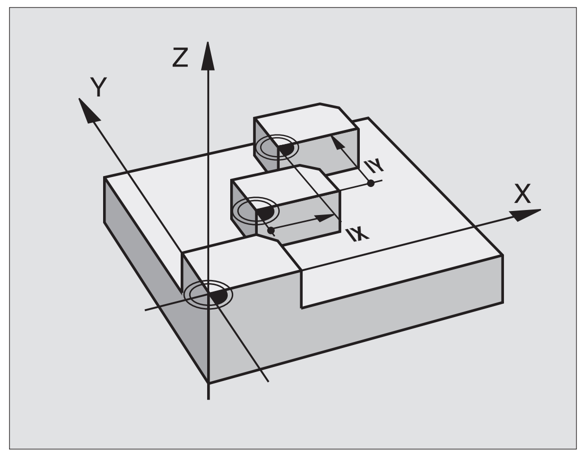
When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.



- **`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)

Datums from a datum table are **always and exclusively** referenced to the current datum (preset).

MP7475, which earlier defined whether datums are referenced to the machine datum or the workpiece datum, now serves only as a safety measure. If MP7475 = 1, the TNC outputs an error message if a datum shift is called from a datum table.

Datum tables from the TNC 4xx whose coordinates are referenced to the machine datum (MP7475 = 1) cannot be used in the iTNC 530.

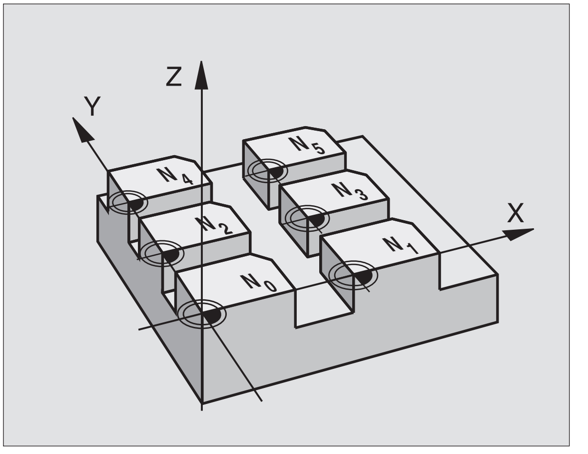
If you are using datum shifts with datum tables, then use the Select Table function to activate the desired datum table from the NC program.

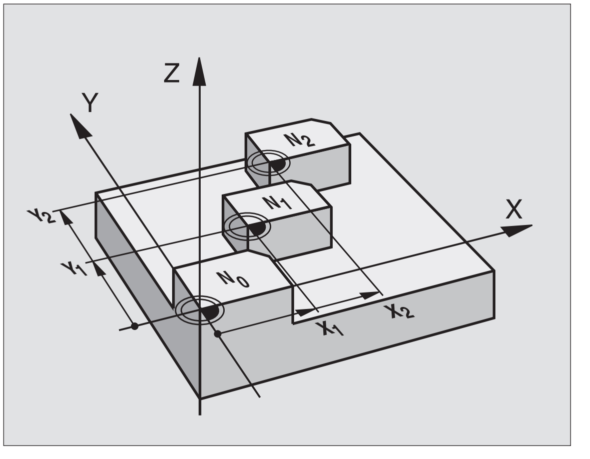
If you work without the Select Table block **%:TAB:**, you must activate the desired datum table before the test run or the program run (This applies also for the programming graphics.):

- 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.



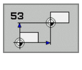


Function

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:

> 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.

PGM CALL

- To select the functions for program call, press the PGM CALL key.

DATUM TABLE

- Press the DATUM TABLE soft key.
- Enter the complete path name of the datum table and confirm your entry with the END key.

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 109).
- 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 the entered line and go to the beginning of the next line	NEXT LINE
Add the entered number of lines (reference points) to the end of the table	APPEND N LINES

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.

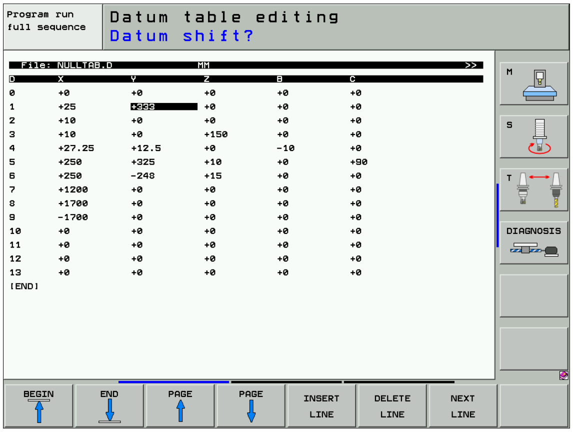
- To enter the values in all axes, press the ALL VALUES soft key.

- 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.

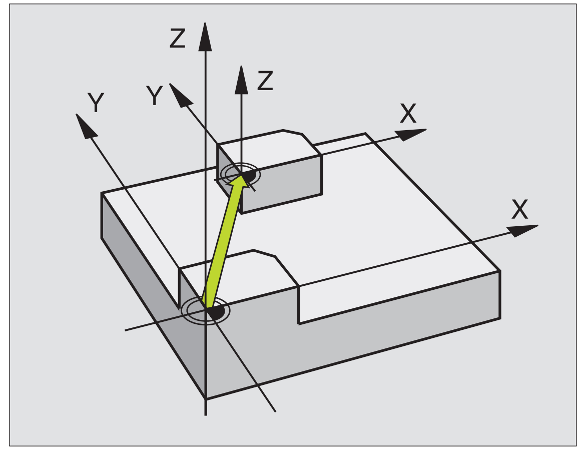
Status displays

In the additional status display, the following data from the datum table are shown (see "Coordinate transformations (TRANS tab)" on page 57):

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

DATUM SETTING (Cycle G247)

With the Cycle DATUM SETTING, you can activate a datum defined in a preset table as the new datum.



Effect

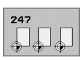
After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

When activating a datum from the preset table, the TNC resets the active datum shift.

The TNC sets the preset only in the axes that are defined with values in the preset table. The datums of axes marked with – remain unchanged.

If you activate preset number 0 (line 0), then you activate the datum that you last set in a manual operating mode.

Cycle G247 is not functional in Test Run mode.



- **Number for datum?:** Enter the number of the datum to be activated from the preset table

Status displays

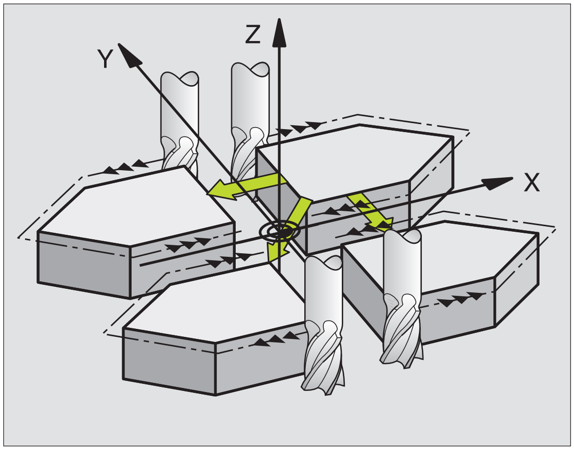
In the status display the TNC shows the active preset number behind the datum symbol

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.



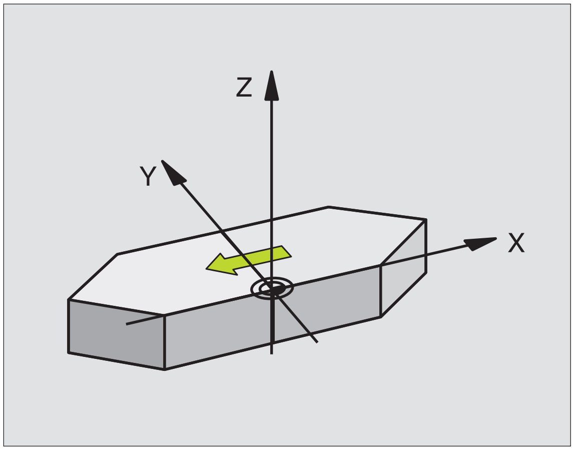
Effect

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.

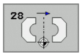
- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

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.



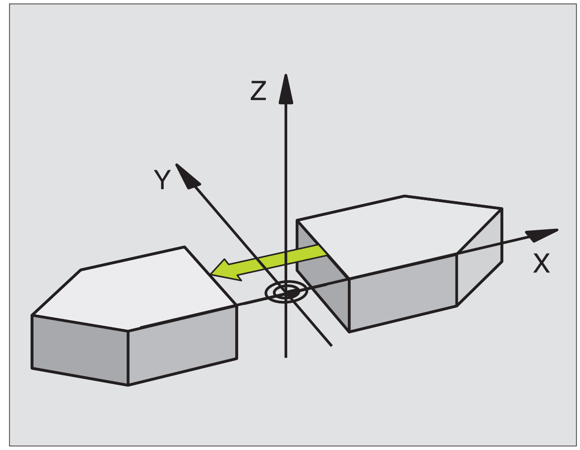
> If you mirror only one axis, the machining direction is reversed for the milling cycles (Cycles 2xx). Exception: Cycle 208, in which the direction defined in the cycle applies.



- **`Mirrored axis?:`** Enter the axis to be mirrored. You can mirror all axes except for the spindle axis—including rotary axes—with the exception of the spindle axis and its associated auxiliary axis. You can enter up to three axes.

Reset

Program the MIRROR IMAGE cycle once again with NO ENT.

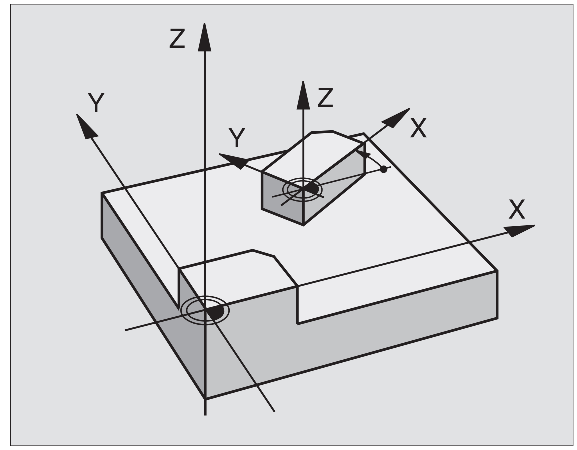


Example: NC block

```
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.



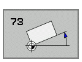
Effect

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

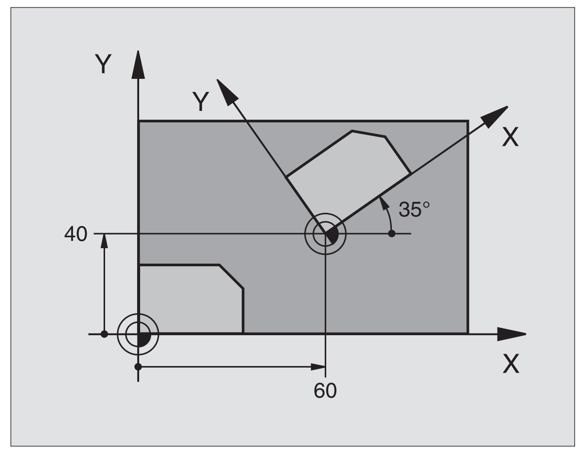
> **Before programming, note the following:**
>
> An active radius compensation is canceled by defining Cycle **G73** and must therefore be reprogrammed, if required.
>
> After defining Cycle **G73,** you must move both axes of the working plane to activate rotation for all axes.



- **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°.

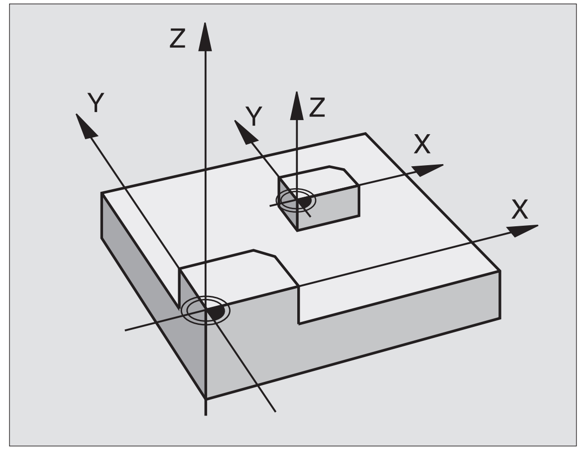


Example: NC block

```
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

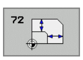
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

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.



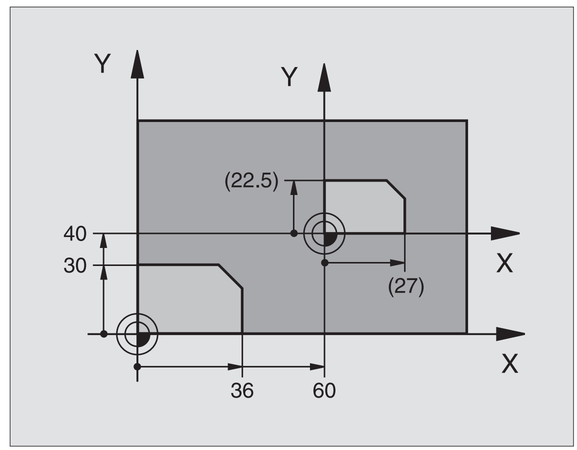
- `Scaling factor?`: Enter the scaling factor F. The TNC multiplies the coordinates and radii by the F factor (as described under "Effect" above).

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.

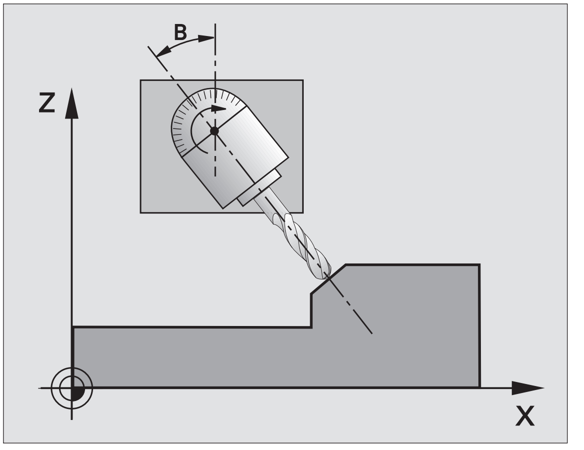


Example: NC blocks

```
N72 G72 F0,750000 *
```

WORKING PLANE (Cycle G80, software option1)

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.



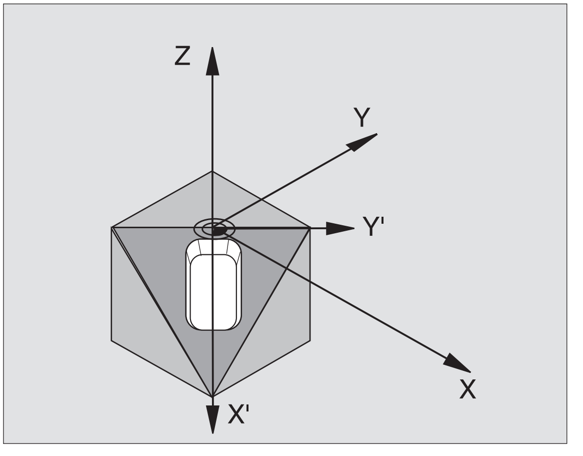
The working plane is always tilted around the active datum.

If you use Cycle 19 when M120 is active, the TNC automatically rescinds the radius compensation, which also rescinds the M120 function.

For fundamentals, see “Tilting the Working Plane (Software Option 1),” page 87. Please read this section completely.

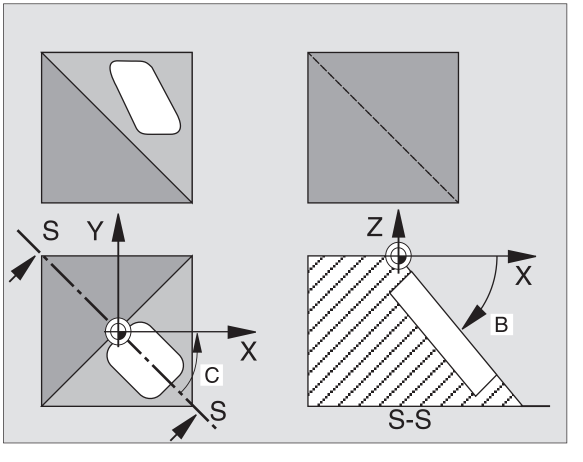
Effect

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 **fixed machine** 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 (Software Option 1),” page 87), 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.

> Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

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 GG40 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 252).

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.
2. Activate tilting function
3. Activate rotation

...

Machining

...

1. Reset the rotation
2. Reset the tilting function
3. 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. For open loop axes, enter the angular values for the rotary axes into the menu.

If the axes are noncontrolled, 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 78).
- 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

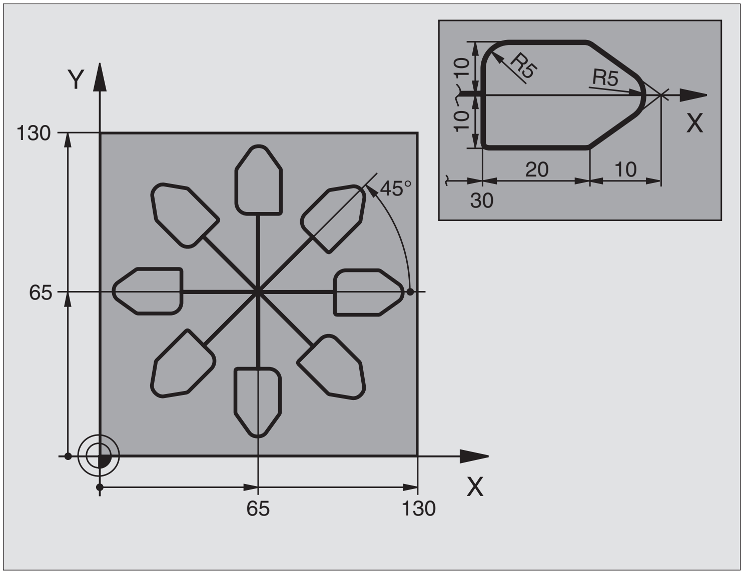
7 Manual Operation mode

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 91).

Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- For subprograms within a subprogram, see "Subprograms," page 491.



`%COTRANS G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	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 S3500 *`	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 *	
N99999999 %COTRANS G71 *	

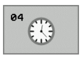
8.10 Special Cycles

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.

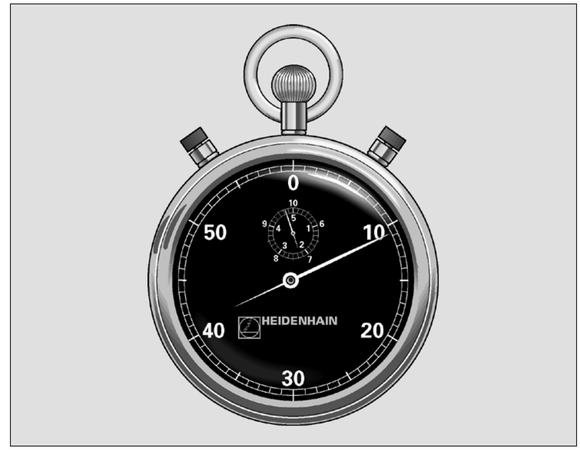
Effect

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



- **`Dwell time in seconds:`** Enter the dwell time in seconds.

Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

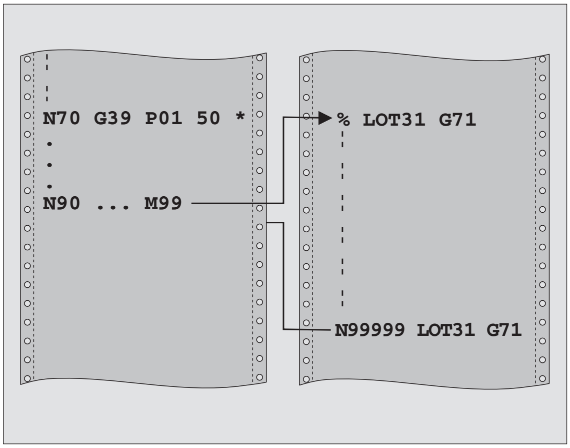


Example: NC block

```
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:

The program you are calling must be stored on the hard disk of your TNC.

If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you 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.

If you want to define an ISO program to be a cycle, enter the file type .I behind the program name.

As a rule, Q parameters are globally effective when called with Cycle G39. So please note that changes to Q parameters in the called program can also influence the calling program.

39 PGM CALL

- **`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 M99 *
```

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

Effect

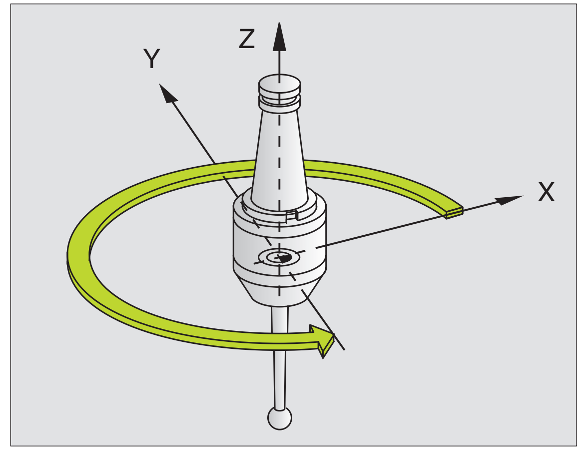
The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

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°

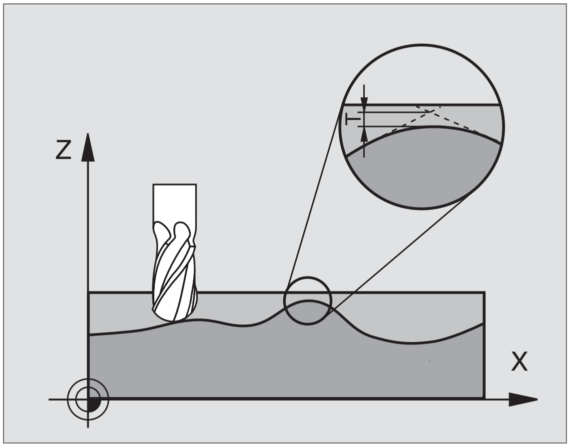


Example: NC block

```
N76 G36 S25 *
```

TOLERANCE (Cycle G62)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.



With the entries in Cycle G62 you can influence the result of HSC machining with respect to accuracy, surface definition and speed, inasmuch as the TNC has been adapted to the machine's characteristics.

The TNC automatically smoothes the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface and therefore reduces wear machine wear. The tolerance defined in the cycle also affects the traverse paths on circular arcs.

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. **Even if the TNC does not move with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the TNC can move the axes.

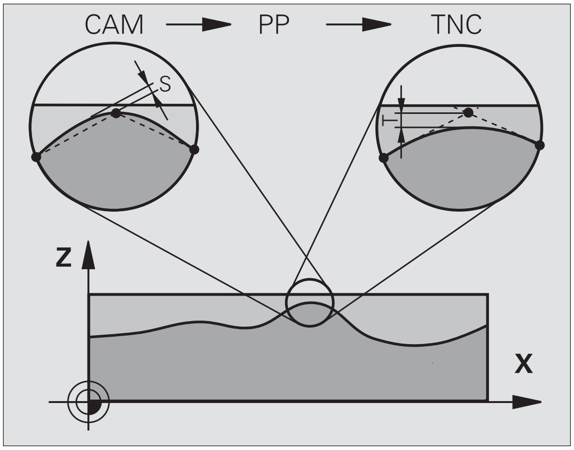
Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error **`tolerance value`** is set in a machine parameter by the machine manufacturer. With **`CYCLE 32,`** you can change the pre-set tolerance value and select different filter settings, provided that your machine manufacturer implements these features.

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the TNC, but by the fact that, in order to machine the contour element transitions very exactly, the TNC might have to drastically reduce the speed.

Influences of the geometry definition in the CAM system

The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system. The maximum point spacing of NC programs generated in a postprocessor (PP) is defined through the chord error. If the chord error is less than or equal to the tolerance value **T** defined in Cycle G62, then the TNC can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing if in Cycle G62 you choose a tolerance value between 110% and 200% of the CAM chord error.



Programming

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.

The TNC resets Cycle G62 if you

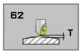
- Redefine it and confirm the dialog question for the **`tolerance value`** with NO ENT.
- Select a new program with the PGM MGT key.

After you have reset Cycle G62, the TNC reactivates the tolerance that was predefined by machine parameter.

In a program with millimeters set as unit of measure, the TNC interprets the entered tolerance value in millimeters. In an inch program it interprets them as inches.

If you transfer a program with Cycle G62 that contains only the cycle parameter **`Tolerance value`** T, the TNC inserts the two remaining parameters with the value 0 if required.

As the tolerance value increases, the diameter of circular movements usually decreases. If the HSC filter is active on your machine (ask your machine manufacturer, if necessary), the circle can also become larger.



- **`Tolerance value:`** Permissible contour deviation in mm (or inches with inch programming)
- **`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. The TNC works with optimal smoothing of the contour points, which results in a reduction of the machining time
- **`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 rotary axis tolerance value. Only the position of the rotary axis with respect to the workpiece surface will change.

The **`P01`** and **`P02`** parameters are only available if on your machine you have software option 2 active (HSC machining).

Example: NC block

```
N78 G62 T0.05 P01 0 P02 5
```

9

Programming: Special Functions

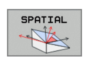
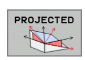
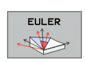
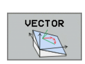
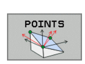
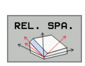
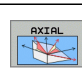
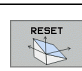
9.1 The PLANE Function: Tilting the Working Plane (Software Option 1)

Introduction

> The machine manufacturer must enable the functions for tilting the working plane!
>
> You can only use the PLANE function on machines which have at least two rotary axes (head and/or table). Exception: The **PLANE AXIAL** can also be used if only a single rotary axis is present or active on your machine

The PLANE function is a powerful function for defining tilted working planes in various manners.

All **PLANE** functions available on the TNC describe the desired working plane independently of the rotary axes actually present on your machine. The following possibilities are available:

Function	Required parameters	Soft key	Page
SPATIAL	Three space angles: **SPA, SPB,** and **SPC**		Page 468
PROJECTED	Two projection angles: **PROPR** and **PROMIN** and a rotation angle **ROT**		Page 470
EULER	Three Euler angles: precession **(EULPR),** nutation **(EULNU)** and rotation **(EULROT)**		Page 472
VECTOR	Norm vector for defining the plane and base vector for defining the direction of the tilted X axis		Page 474
POINTS	Coordinates of any three points in the plane to be tilted		Page 476
RELATIVE	Single, incrementally effective spatial angle		Page 478
AXIAL	Up to three absolute or incremental axis angles **A**, **B**, **C**		Page 480
RESET	Reset the PLANE function		Page 467

In order to make the differences between each definition possibility more clear even before selecting the function, you can start an animated sequence via soft key.

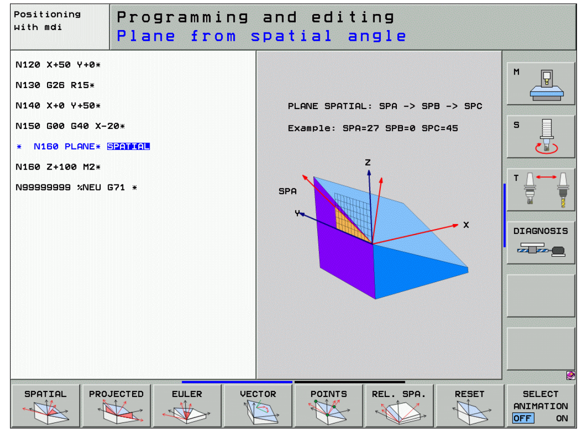
The parameter definition of the **PLANE** function is separated into two parts:

- The geometric definition of the plane, which is different for each of the available **PLANE** functions.
- The positioning behavior of the **PLANE** function, which is independent of the plane definition and is identical for all **PLANE** functions (see "Specifying the Positioning Behavior of the PLANE Function" on page 482).

The actual-position-capture function is not possible with an active tilted working plane.

If you use the **PLANE** function when M120 is active, the TNC automatically rescinds the radius compensation, which also rescinds the M120 function.

Define the PLANE function



SPEC FCT

- Show the soft-key row with special functions

SPECIAL TNC FUNCTIONS

- Select special TNC functions: Press the SPECIAL TNC FUNCTIONS soft key.

TILT MACHINING PLANE

- Select the **PLANE** function: Press the TILT MACHINING PLANE soft key: The TNC displays the available definition possibilities in the soft-key row.

Selecting the function while animation is active

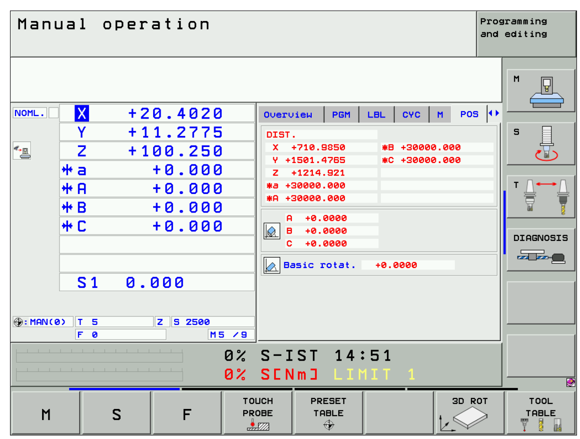
- Activate animation: Set the SELECT ANIMATION ON/OFF soft key to ON.
- Start an animation for one of the definition possibilities: Press one of the available soft keys. The TNC highlights the soft key with a different color and begins the appropriate animation.
- To assume the currently active function: Press the ENT key or press the soft key of the active function again. The TNC continues the dialog and requests the required parameters.

Selecting the function while animation is inactive

- Select the desired function directly via soft key. The TNC continues the dialog and requests the required parameters.

Position display

As soon as a **PLANE** function is active, the TNC shows the calculated spatial angle in the additional status display (see figure). As a rule, the TNC internally always calculates with spatial angles, independent of which **PLANE** function is active.



Reset the PLANE function

SPEC FCT

▶ Show the soft-key row with special functions

SPECIAL TNC FUNCTIONS

▶ Select special TNC functions: Press the SPECIAL TNC FUNCTIONS soft key.

TILT MACHINING PLANE

▶ Select the PLANE function: Press the TILT MACHINING PLANE soft key: The TNC displays the available definition possibilities in the soft-key row.

RESET

▶ Select the Reset function. This internally resets the **PLANE** function, but does not change the current axis positions.

MOVE

▶ Specify whether the TNC should automatically move the rotary axes to the default setting **(MOVE** or **TURN)** or not **(STAY)**. (see "Automatic positioning: MOVE/TURN/STAY (entry is mandatory)" on page 482)

END

▶ To terminate entry, press the END key.

The **PLANE RESET** function resets the current **PLANE** function—or an active Cycle 19—completely (angles = 0 and function is inactive). It does not need to be defined more than once.

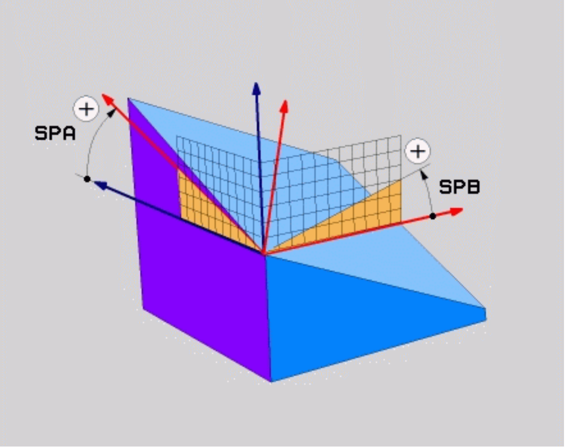
Example: NC block

```
N25 PLANE RESET MOVE SET UP 50 F1000 *
```

9.2 Defining the Machining Plane with Space Angles: PLANE SPATIAL

Function

Spatial angles define a machining plane through up to three **rotations around the fixed machine coordinate system.** The sequence of rotations is firmly specified: first around the A axis, then B, and then C (the function corresponds to Cycle 19, if the entries in Cycle 19 are set to space angles).



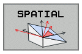
Before programming, note the following:

You must always define the three space angles **SPA, SPB,** and **SPC,** even if one of them = 0.

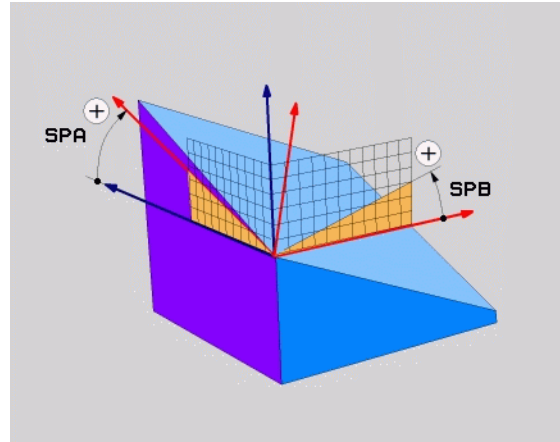
The sequence of the rotations described above is independent of the active tool axis.

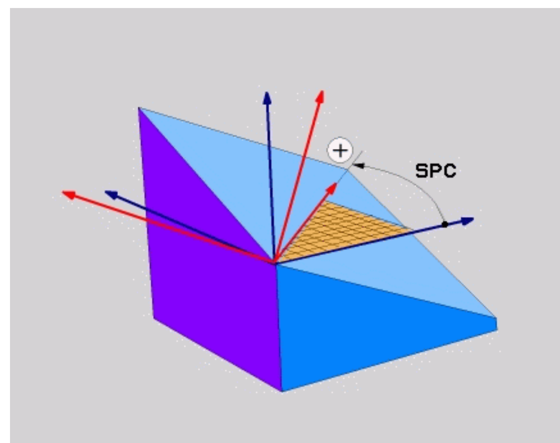
Parameter description for the positioning behavior: See "Specifying the Positioning Behavior of the PLANE Function," page 482.

Input parameters



- **Space angle A?:** Rotational angle **SPA** around the fixed machine axis X (see figure at top right). Input range: from –359.9999° to +359.9999°
- **Spatial angle B?:** Rotational angle **SPB** around the fixed machine axis Y (see figure at top right). Input range: from –359.9999° to +359.9999°
- **Spatial angle C?:** Rotational angle **SPC** around the fixed machine axis Z (see figure at center right). Input range: from –359.9999° to +359.9999°
- Continue with the positioning properties (see "Specifying the Positioning Behavior of the PLANE Function" on page 482).





Abbreviations used

Abbreviation	Meaning
SPATIAL	**Spatial** = in space
SPA	**Sp**atial **A:** rotation about the X axis
SPB	**Sp**atial **B:** rotation about the Y axis
SPC	**Sp**atial **C:** rotation about the Z axis

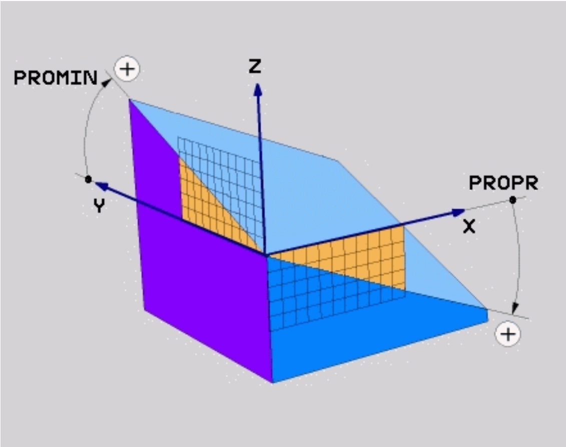
Example: NC block

```
N50 PLANE SPATIAL SPA+27 SPB+0 SPC+45 ...
```

9.3 Defining the Machining Plane with Projection Angles: PROJECTED PLANE

Function

Projection angles define a machining plane through the entry of two angles that you determine by projecting the first coordinate plane (Z/X plane with tool axis Z) and the second coordinate plane (Y/Z with tool axis Z) onto the machining plane to be defined.

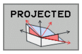


Before programming, note the following:

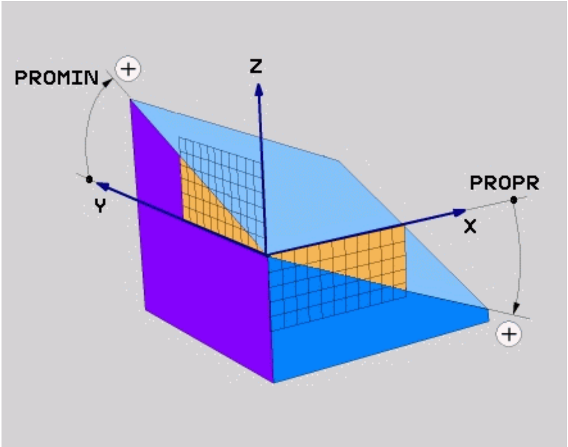
You can only use projection angles if a rectangular cuboid is to be machined. Otherwise distortions could occur on the workpiece.

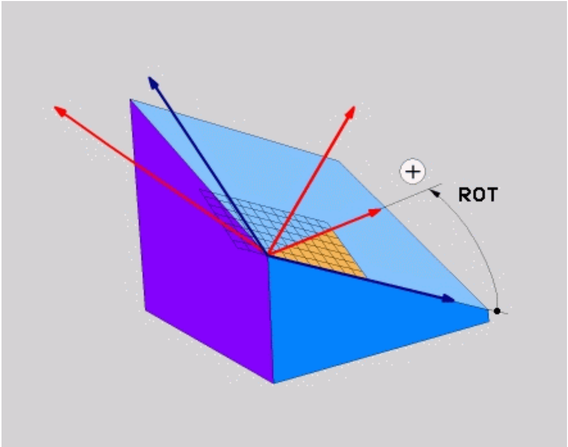
Parameter description for the positioning behavior: See "Specifying the Positioning Behavior of the PLANE Function," page 482.

Input parameters



- **Proj. angle 1st coordinate plane?:** Projected angle of the tilted machining plane in the 1st coordinate plane of the fixed machine coordinate system (Z/X for tool axis Z, see figure at top right). Input range: from –89.9999° to +89.9999°. The 0° axis is the principal axis of the active machining plane (X for tool axis Z. See figure at top right for positive direction).
- **Proj. angle 2nd coordinate plane?:** Projected angle in the 2nd coordinate plane of the fixed machine coordinate system (Y/Z for tool axis Z, see figure at top right). Input range: from -89.9999° to +89.9999°. The 0° axis is the minor axis of the active machining plane (Y for tool axis Z).
- **ROT angle of the tilted plane?:** Rotation of the tilted coordinate system around the tilted tool axis (corresponds to a rotation with Cycle 10 ROTATION). The rotation angle is used to simply specify the direction of the principal axis of the machining plane (X for tool axis Z, Z for tool axis Y; see figure at bottom right). Input range: from 0° to +360°.
- Continue with the positioning properties (see "Specifying the Positioning Behavior of the PLANE Function" on page 482).





Abbreviations used

Abbreviation	Meaning
PROJECTED	Projected
PROPR	**Pr**incipal plane
PROMIN	**Min**or plane
PROROT	**Rot**ation

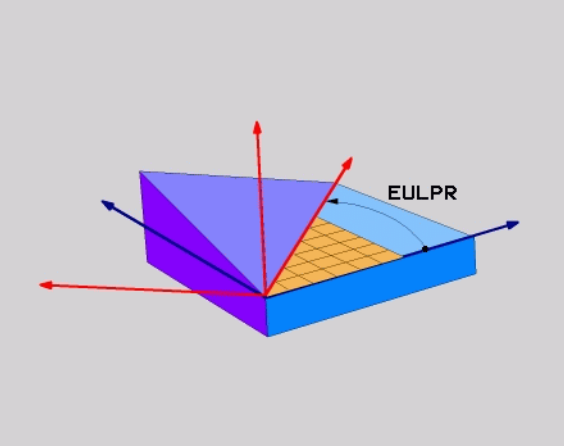
Example: NC block

```
N50 PLANE PROJECTED PROPR+24 PROMIN+24 PRO
ROT+30 ...
```

9.4 Defining the Machining Plane with Euler Angles: PLANE EULER

Function

Euler angles define a machining plane through up to three **rotations about the respectively tilted coordinate system.** The Swiss mathematician Leonhard Euler defined these angles. When applied to the machine coordinate system, they have the following meanings:



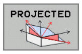
Precession angle **EULPR**	Rotation of the coordinate system around the Z axis
Nutation angle **EULNU**	Rotation of the coordinate system around the X axis already shifted by the precession angle
Rotation angle **EULROT**	Rotation of the tilted machining plane around the tilted Z axis

Before programming, note the following:

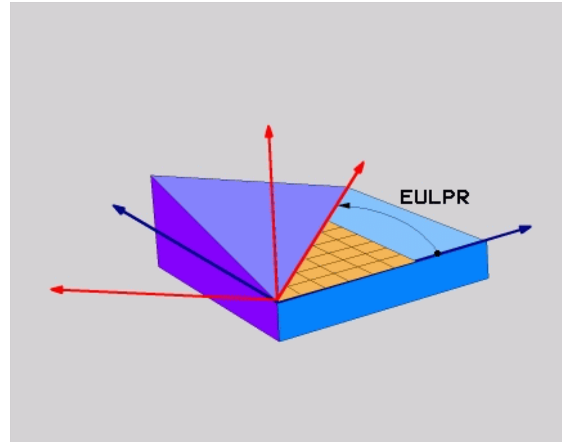
The sequence of the rotations described above is independent of the active tool axis.

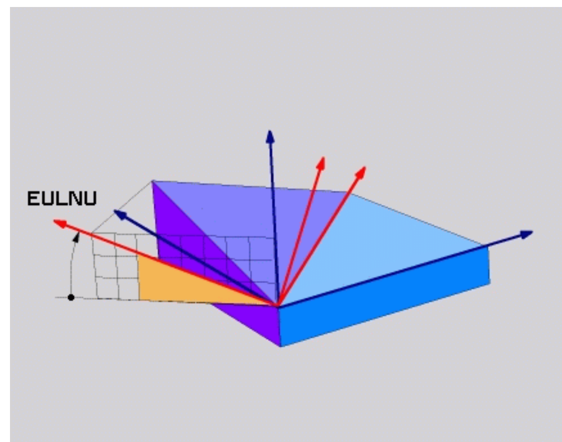
Parameter description for the positioning behavior: See "Specifying the Positioning Behavior of the PLANE Function," page 482.

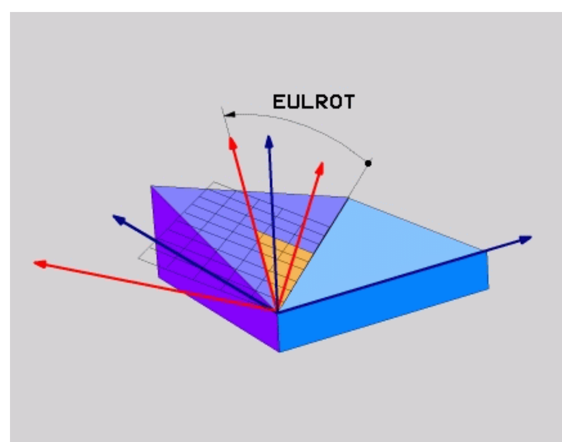
Input parameters



- **`Rot. angle main coordinate plane?:`** Rotary angle **`EULPR`** around the Z axis (see figure at top right). Note:
  - Input range: –180.0000° to 180.0000°
  - The 0° axis is the X axis.
- **`Tilting angle tool axis?:`** Tilting angle **`EULNUT`** of the coordinate system around the X axis shifted by the precession angle (see figure at center right). Note:
  - Input range: from 0° to 180.0000°
  - The 0° axis is the Z axis.
- **`ROT angle of the tilted plane?`**: Rotation **`EULROT`** of the tilted coordinate system around the tilted Z axis (corresponds to a rotation with Cycle 10 ROTATION). Use the rotation angle to simply define the direction of the X axis in the tilted machining plane (see figure at bottom right). Note:
  - Input range: from 0° to 360.0000°
  - The 0° axis is the X axis.
- Continue with the positioning properties (see “Specifying the Positioning Behavior of the PLANE Function” on page 482).







NC block

```
N50 PLANE EULER EULPR45 EULNU20 EULROT22 ...
```

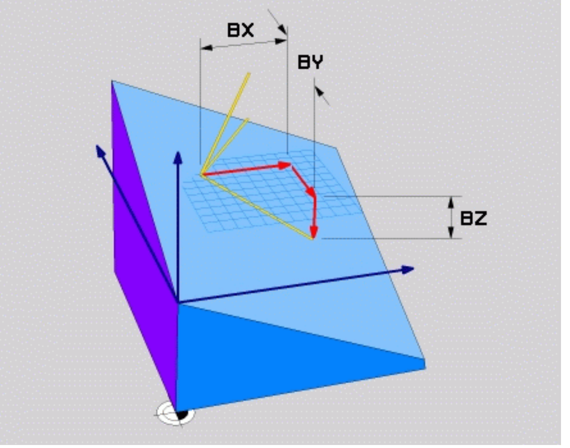
Abbreviations used

Abbreviation	Meaning
EULER	Swiss mathematician who defined these angles
EULPR	**Pr**ecession angle: angle describing the rotation of the coordinate system around the Z axis
EULNU	**Nu**tation angle: angle describing the rotation of the coordinate system around the X axis shifted by the precession angle
EULROT	**Rot**ation angle: angle describing the rotation of the tilted machining plane around the tilted Z axis

9.5 Defining the Machining Plane with Two Vectors: VECTOR PLANE

Function

You can use the definition of a machining plane via **two vectors** if your CAD system can calculate the base vector and normal vector of the tilted machining plane. A normalized input is not necessary. The TNC calculates the normal, so you can enter values between –99.999999 and +99.999999.



The base vector required for the definition of the machining plane is defined by the components **BX, BY** and **BZ** (see figure at right). The normal vector is defined by the components **NX, NY** and **NZ.**

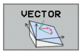
The base vector defines the direction of the X axis in the tilted machining plane, and the normal vector determines the direction of the machining plane, and at the same time is perpendicular to it.

Before programming, note the following:

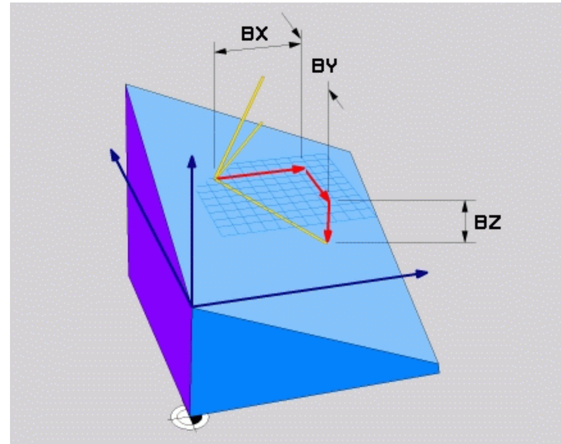
The TNC calculates standardized vectors from the values you enter.

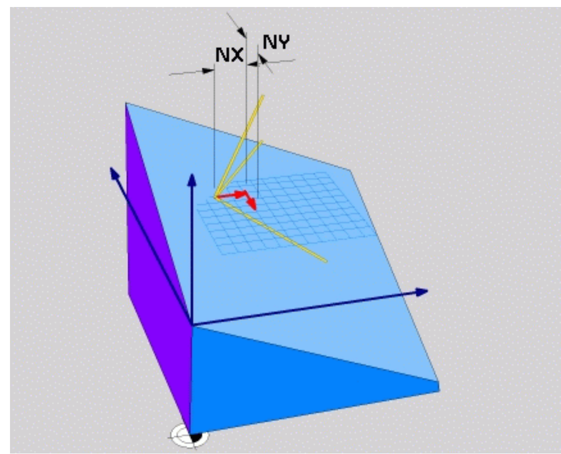
Parameter description for the positioning behavior: See "Specifying the Positioning Behavior of the PLANE Function," page 482.

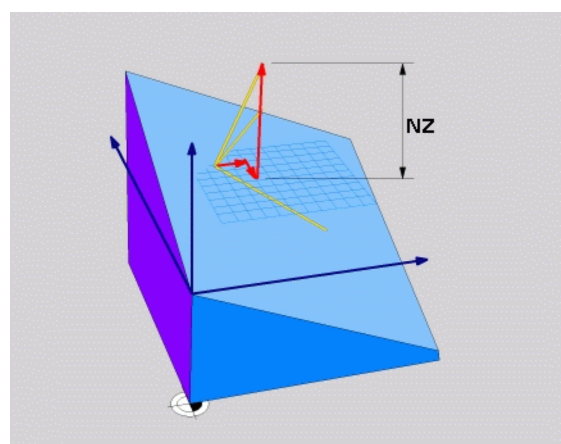
Input parameters



- **X component of base vector?:** X component **BX** of the base vector B (see figure at top right). Input range: -99.9999999 to +99.9999999
- **Y component of base vector?:** Y component **BY** of the base vector B (see figure at top right). Input range: -99.9999999 to +99.9999999
- **Z component of base vector?:** Z component **BZ** of the base vector B (see figure at top right). Input range: -99.9999999 to +99.9999999
- **X component of normal vector?:** X component **NX** of the normal vector N (see figure at center right). Input range: -99.9999999 to +99.9999999
- **Y component of normal vector?:** Y component **NY** of the normal vector N (see figure at center right). Input range: -99.9999999 to +99.9999999
- **Z component of normal vector?:** Z component **NZ** of the normal vector N (see figure at lower right). Input range: -99.9999999 to +99.9999999
- Continue with the positioning properties (see "Specifying the Positioning Behavior of the PLANE Function" on page 482).







NC block

```
N50 PLANE VECTOR BX0.8 BY-0.4 BZ-
0.4472 NX0.2 NY0.2 NZ0.9592 ...
```

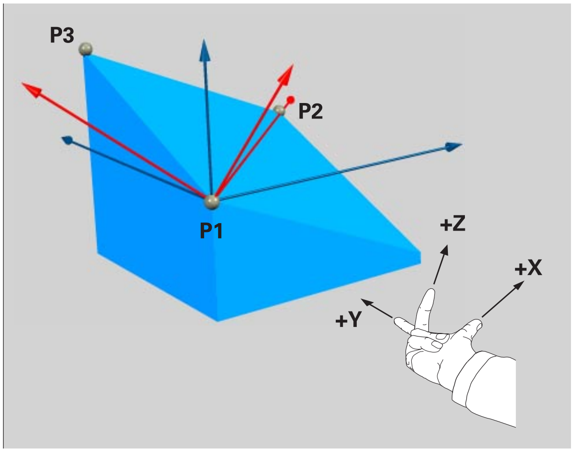
Abbreviations used

Abbreviation	Meaning
VECTOR	Vector
BX, BY, BZ	**B**ase vector: **X, Y** and **Z** components
NX, NY, NZ	**N**ormal vector: **X, Y** and **Z** components

9.6 Defining the Machining Plane via Three Points: POINTS PLANE

Function

A machining plane can be uniquely defined by entering **any three points P1 to P3 in this plane.** This possibility is realized in the **POINTS PLANE** function.



Before programming, note the following:

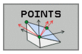
The connection from Point 1 to Point 2 determines the direction of the tilted principal axis (X for tool axis Z).

The direction of the tilted tool axis is determined by the position of Point 3 relative to the connecting line between Point 1 and Point 2. Use the right-hand rule (thumb = X axis, index finger = Y axis, middle finger = Z axis (see figure at right)) to remember: thumb (X axis) points from Point 1 to Point 2, index finger (Y axis) points parallel to the tilted Y axis in the direction of Point 3. Then the middle finger points in the direction of the tilted tool axis.

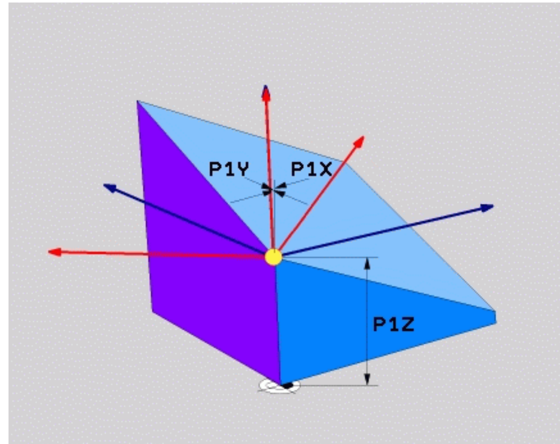
The three points define the slope of the plane. The position of the active datum is not changed by the TNC.

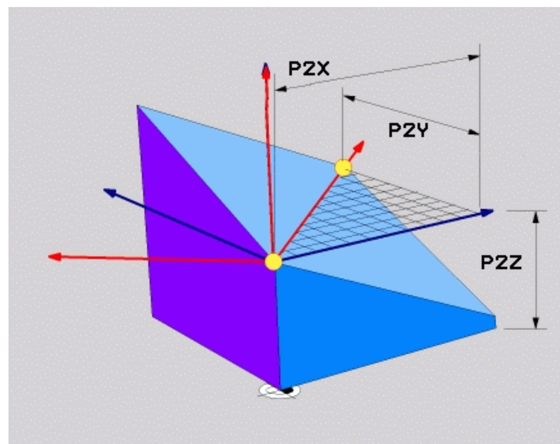
Parameter description for the positioning behavior: See "Specifying the Positioning Behavior of the PLANE Function," page 482.

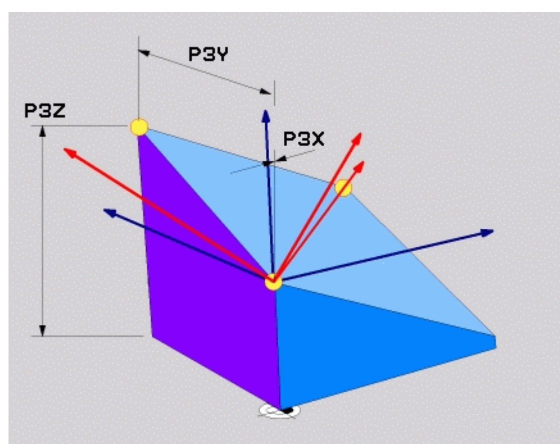
Input parameters



- **`X coordinate of 1st plane point?:`** X coordinate **`P1X`** of the 1st plane point (see figure at top right).
- **`Y coordinate of 1st plane point?:`** Y coordinate **`P1Y`** of the 1st plane point (see figure at top right).
- **`Z coordinate of 1st plane point?:`** Z coordinate **`P1Z`** of the 1st plane point (see figure at top right).
- **`X coordinate of 2nd plane point?:`** X coordinate **`P2X`** of the 2nd plane point (see figure at center right).
- **`Y coordinate of 2nd plane point?:`** Y coordinate **`P2Y`** of the 2nd plane point (see figure at center right).
- **`Z coordinate of 2nd plane point?:`** Z coordinate **`P2Z`** of the 2nd plane point (see figure at center right).
- **`X coordinate of 3rd plane point?:`** X coordinate **`P3X`** of the 3rd plane point (see figure at bottom right).
- **`Y coordinate of 3rd plane point?:`** Y coordinate **`P3Y`** of the 3rd plane point (see figure at bottom right).
- **`Z coordinate of 3rd plane point?:`** Z coordinate **`P3Z`** of the 3rd plane point (see figure at bottom right).
- Continue with the positioning properties (see “Specifying the Positioning Behavior of the PLANE Function” on page 482).







NC block

```
N50 PLANE POINTS P1X+0 P1Y+0 P1Z+20 P2X+30 P2Y+31 P2Z+20
P3X+0 P3Y+41 P3Z+32.5 ...
```

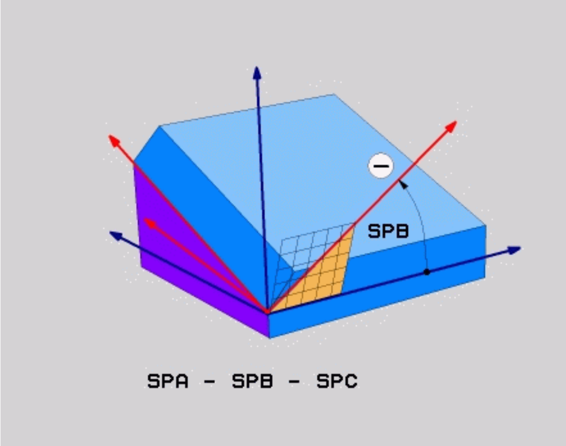
Abbreviations used

Abbreviation	Meaning
`POINTS`	Points

9.7 Defining the Machining Plane with a Single, Incremental Space Angle: PLANE RELATIVE

Function

Use the incremental space angle when an already active tilted machining plane is to be tilted by **another rotation.** Example: machining a 45° chamfer on a tilted plane.



Before programming, note the following:

The defined angle always applies to the active machining plane, no matter which function you used to activate it.

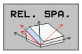
You can program any number of **PLANE RELATIV** functions in a row.

If you want to return to the machining plane that was active before the **PLANE RELATIV** function, redefine the **PLANE RELATIV** function with the same angle but with the opposite algebraic sign.

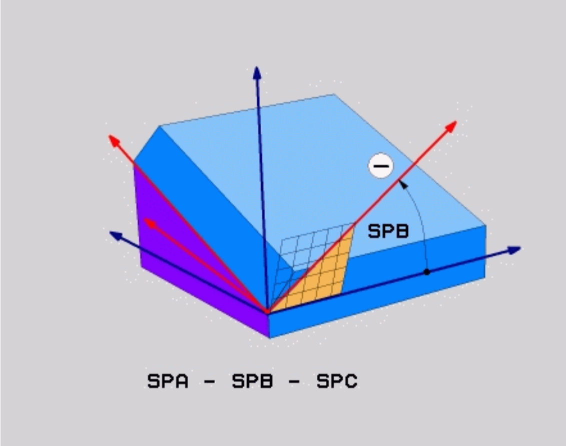
If you use the **PLANE RELATIV** function on an untilted machining plane, then you simply rotate the untilted plane about the space angle defined in the **PLANE** function.

Parameter description for the positioning behavior: See "Specifying the Positioning Behavior of the PLANE Function," page 482.

Input parameters



- **`Incremental angle?:`** Spatial angle about which the active machining plane is to be rotated additionally (see figure at right). Use a soft key to select the axis to be rotated about. Input range: –359.9999° to +359.9999°
- Continue with the positioning properties (see "Specifying the Positioning Behavior of the PLANE Function" on page 482).



Example: NC block

```
N50 PLANE RELATIV SPB-45 ...
```

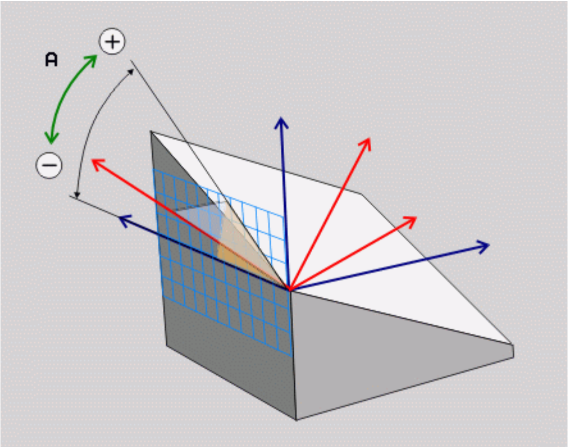
Abbreviations used

Abbreviation	Meaning
`RELATIVE`	Relative

9.8 Tilting the working plane through axis angle: PLANE AXIAL (FCL 3 function)

Function

The **PLANE AXIAL** function defines both the position of the working plane and the nominal coordinates of the rotary axes. This function is particularly easy to use on machines with Cartesian coordinates and with kinematic structures in which only one rotary axis is active.



The **PLANE AXIAL** can also be used if you have only one rotary axis active on your machine

You can use the **PLANE RELATIV** function after **PLANE AXIAL** if your machine allows spatial angle definitions. The machine tool manual provides further information.

Before programming, note the following:

Enter only axis angles that actually exist on your machine. Otherwise the TNC generates an error message.

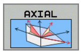
Rotary axis coordinates defined with **PLANE AXIAL** are modally effective. Successive definitions therefore build on each other. Incremental input is allowed.

To reset the **PLANE AXIS** function, use **PLANE RESET.** Resetting by entering 0 does not deactivate **PLANE AXIAL.**

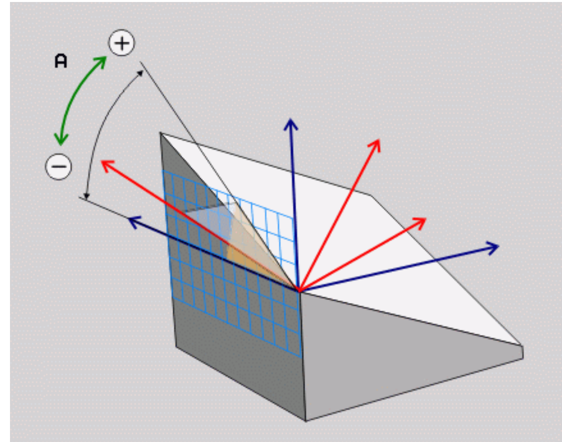
SEQ, TABLE ROT and **COORD ROT** have no function in conjunction with **PLANE AXIS.**

Parameter description for the positioning behavior: See “Specifying the Positioning Behavior of the PLANE Function,” page 482.

Input parameters



- **`Axis angle A?:`** Axis angle **to which** the A axis is to be moved. If entered incrementally, it is the angle, **by which** the A axis is to be moved from its current position. Input range: -99999.9999° to +99999.9999°
- **`Axis angle B?:`** Axis angle **to which** the B axis is to be moved. If entered incrementally, it is the angle, **by which** the B axis is to be moved from its current position. Input range: -99999.9999° to +99999.9999°
- **`Axis angle C?:`** Axis angle **to which** the C axis is to be moved. If entered incrementally, it is the angle, **by which** the C axis is to be moved from its current position. Input range: -99999.9999° to +99999.9999°
- Continue with the positioning properties (see "Specifying the Positioning Behavior of the PLANE Function" on page 482).



Abbreviations used

Abbreviation	Meaning
`AXIAL`	Axis-shaped

Example: NC block

```
5 PLANE AXIAL B-45 .....
```

9.9 Specifying the Positioning Behavior of the PLANE Function

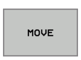
Overview

Independently of which PLANE function you use to define the tilted machining plane, the following functions are always available for the positioning behavior:

- Automatic positioning
- Selection of alternate tilting possibilities
- Selection of the type of transformation

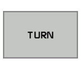
Automatic positioning: `MOVE/TURN/STAY` (entry is mandatory)

After you have entered all parameters for the plane definition, you must specify how the rotary axes will be positioned to the calculated axis values:

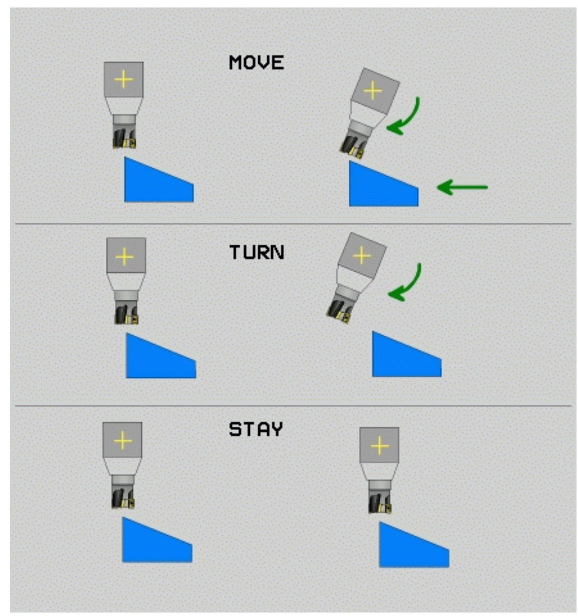


- The PLANE function is to automatically position the rotary axes to the calculated position values. The position of the tool relative to the workpiece is to remain the same. The TNC carries out a compensating motion in the linear axes.

- The PLANE function is to automatically position the rotary axes to the calculated position values, but only the rotary axes are positioned. The TNC does **not** carry out a compensating motion in the linear axes.



- You will position the rotary axes later in a separate positioning block.



If you have selected the **`MOVE`** option (**`PLANE`** function is to position the axes automatically), the following two parameters must still be defined: **`Dist. tool tip – center of rot.`** and **`Feed rate? F=`**. If you have selected the **`TURN`** option (**`PLANE`** function is to position the axes automatically without any compensating movement), the following parameter must still be defined: **`Feed rate? F=`**.

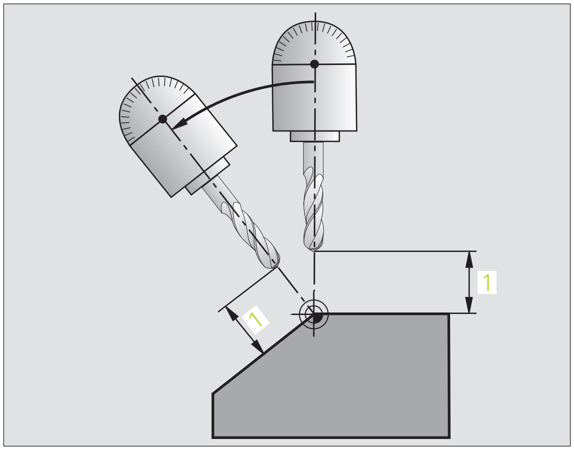
> If you use **`PLANE AXIAL`** together with **`STAY,`** you have to position the rotary axes in a separated block after the **`PLANE`** function.

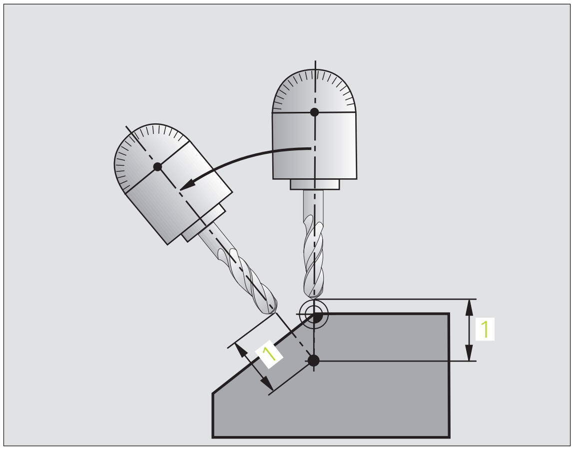
- **`Dist. tool tip – center of rot.`** (incremental): The TNC tilts the tool (or table) relative to the tool tip. The **`SETUP`** parameter shifts the center of rotation of the positioning movement relative to the current position of the tool tip.

Note:

- If the tool is already at the given distance to the workpiece before positioning, then relatively speaking the tool is at the same position after positioning (see figure at center right, **1** = SET UP).
- If the tool is not at the given distance to the workpiece before positioning, then relatively speaking the tool is offset from the original position after positioning (see figure at bottom right, **1** = SET UP).

- **`Feed rate ? F=`**: Contour speed at which the tool should be positioned.





Positioning the rotary axes in a separate block

Proceed as follows if you want to position the rotary axes in a separate positioning block (option **STAY** selected):

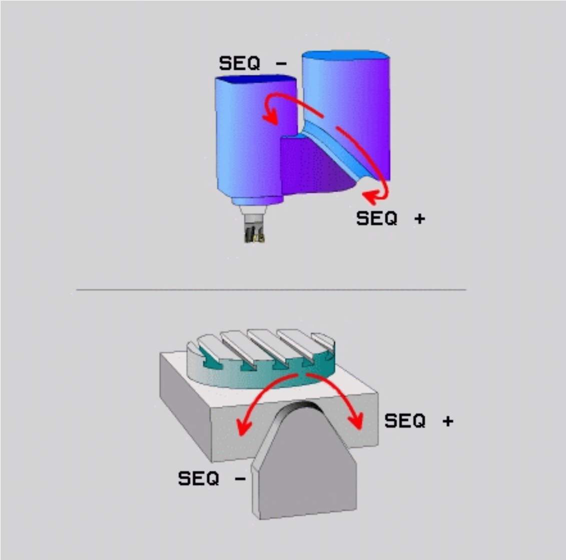
> Pre-position the tool to a position where there is no danger of collision with the workpiece (clamping devices) during positioning.

- Select any **PLANE** function, and define automatic positioning with the **STAY** option. During program execution the TNC calculates the position values of the rotary axes present on the machine, and stores them in the system parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis).
- Define the positioning block with the angular values calculated by the TNC.

NC example blocks: Position a machine with a rotary table C and an tilting table A to a space angle of B+45°.

`...`	
`N120 G00 G40 Z+250 *`	Position at clearance height
`N130 PLANE SPATIAL SPA+0 SPB+45 SPC+0 STAY *`	Define and activate the PLANE function
`N140 G01 F2000 A+Q120 C+Q122 *`	Position the rotary axis with the values calculated by the TNC
`...`	Define machining in the tilted working plane

Selection of alternate tilting possibilities: SEQ +/– (entry optional)



The position you define for the machining plane is used by the TNC to calculate the appropriate positioning of the rotary axes present on the machine. In general there are always two solution possibilities.

Use the **SEQ** switch to specify which possibility the TNC should use:

- **SEQ+** positions the master axis so that it assumes a positive angle. The master axis is the 2nd rotary axis from the table, or the 1st axis from the tool (depending on the machine configuration (see figure at top right)).
- **SEQ–** positions the master axis so that it assumes a negative angle.

If the solution you chose with **SEQ** is not within the machine's range of traverse, the TNC displays the **Entered angle not permitted** error message.

> When the **PLANE AXIS** function is used, the **PLANE RESET** switch is nonfunctional.

If you do not define **SEQ,** the TNC determines the solution as follows:

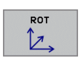
1. The TNC first checks whether both solution possibilities are within the traverse range of the rotary axes.
2. If they are, then the TNC selects the shortest possible solution.
3. If only one solution is within the traverse range, the TNC selects this solution.
4. If neither solution is within the traverse range, the TNC displays the **Entered angle not permitted** error message.

Example for a machine with a rotary table C and an tilting table A.
Programmed function: **PLANE SPATIAL SPA+0 SPB+45 SPC+0**

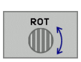
Limit switch	Starting position	SEQ	Resulting axis position
None	A+0, C+0	not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A–45, C–90
None	A+0, C-105	not prog.	A–45, C–90
None	A+0, C-105	+	A+45, C+90
None	A+0, C-105	–	A–45, C–90
–90 < A < +10	A+0, C+0	not prog.	A–45, C–90
–90 < A < +10	A+0, C+0	+	Error message
None	A+0, C-135	+	A+45, C+90

Selecting the type of transformation (entry optional)

On machines with C-rotary tables, a function is available for specifying the type of transformation:

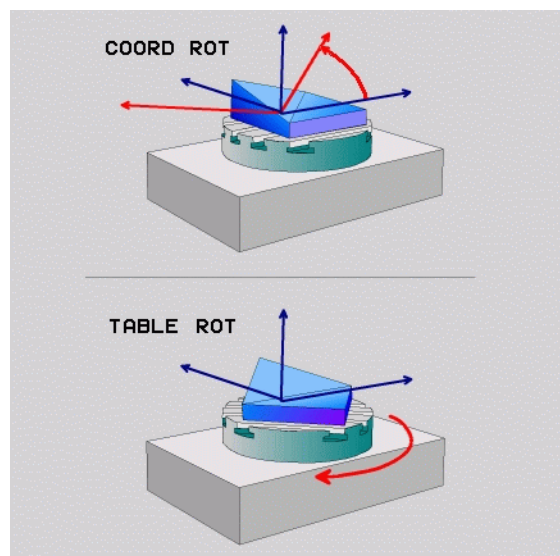


- **COORD ROT** specifies that the PLANE function should only rotate the coordinate system to the defined tilting angle. The rotary table is not moved; the compensation is purely mathematical.



- **TABLE ROT** specifies that the PLANE function should position the rotary table to the defined tilting angle. Compensation results from rotating the workpiece.

When the **PLANE AXIS** function is used, **COORD ROT** and **TABLE ROT** are nonfunctional.



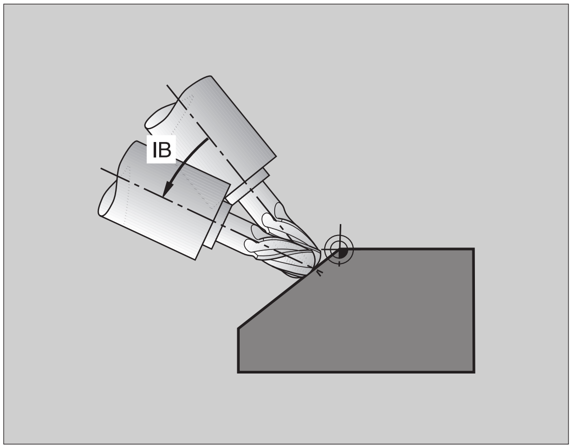
9.10 Inclined-Tool Machining in the Tilted Plane

Function

In combination with M128 and the new **PLANE** functions, **inclined-tool machining** in a tilted machining plane is now possible. Two possibilities are available for definition:

- Inclined-tool machining via incremental traverse of a rotary axis
- Inclined-tool machining via normal vectors (only conversational)

> Inclined-tool machining in a tilted machining plane only functions with spherical cutters.
>
> With 45° swivel heads and tilting tables you can also define the incline angle as a space angle. Use the **TCPM FUNCTION** (only conversational).



Inclined-tool machining via incremental traverse of a rotary axis

- Retract the tool
- Activate M128
- Define any PLANE function; consider the positioning behavior
- Via an L block, traverse to the desired incline angle in the appropriate axis incrementally

Example NC blocks:

...	
N120 G00 G40 Z+50 M128 *	Position at clearance height, activate M128
N130 PLANE SPATIAL SPA+0 SPB-45 SPC+0 MOVE ABST50 F1000 *	Define and activate the PLANE function
N140 G01 G91 F1000 B-17 *	Set the incline angle
...	Define machining in the tilted working plane

10

Programming: Subprograms and Program Section Repeats

10.1 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.

Label

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 999 or by a name you define. Each LABEL number or LABEL name can be set only once in the program with **G98.** The number of label names you can enter is only limited by the internal memory.

> If a label name or number 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.

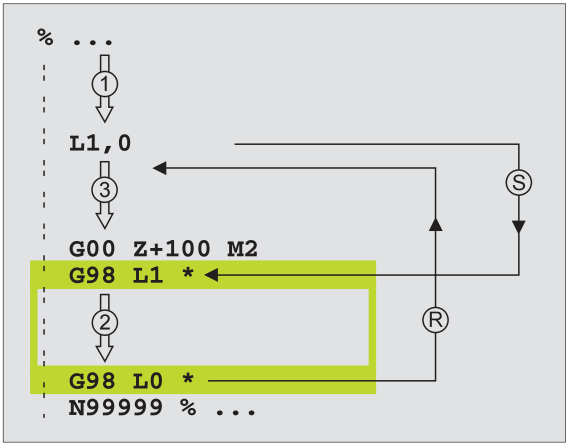
10.2 Subprograms

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 M02 or M30).
- If subprograms are located before the block with M02 or M30, they will be executed at least once even if they are not called.

Programming a subprogram

LBL SET

- To mark the beginning, press the LBL SET key.
- Enter the subprogram number and confirm with the END key. If you want to use a label name, press the key " to switch to text entry.
- 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. If you want to use a label name, press the key " to switch to text entry.
- **Repeat REP:** Enter "**.0**", then confirm with the ENT key.

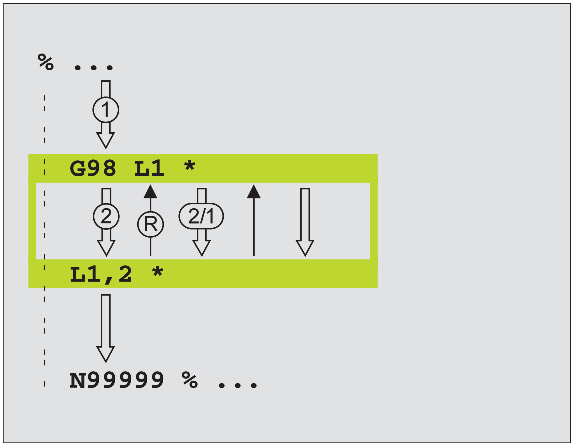
L0.0 is not permitted, as it corresponds to the program end call.

10.3 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. If you want to use a label name, press the key " to switch to text entry.

Calling a program section repeat

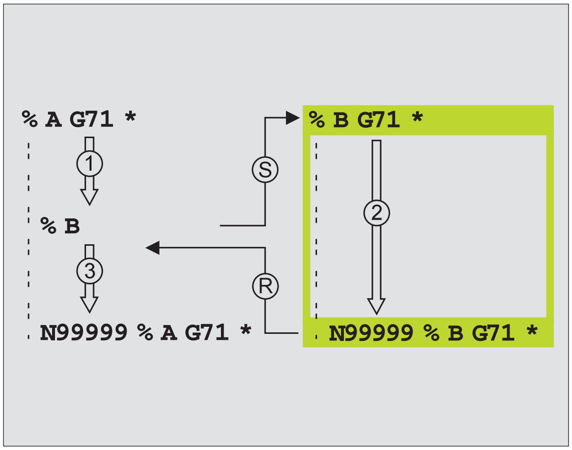
LBL CALL

- Press the LBL CALL key.
- **Label number:** Enter the label number of the subprogram to be called, then confirm with the ENT key. If you want to use a label name, press the key " to switch to text entry.
- **Repeat REP:** Enter the number of repeats, then confirm with the ENT key.

10.4 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 M02 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.

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\SCHRUPP\PGM1.H**

If you want to call a conversational dialog program, enter the file type .H behind the program name.

You can also call a program with Cycle **G39.**

As a rule, Q parameters are effective globally with a **% (PGM CALL).** So please note that changes to Q parameters in the called program can also influence the calling program.

Coordinate transformations that you define in the called program remain in effect for the calling program too, unless you reset them. The setting of machine parameter MP7300 has no influence on this.

10.5 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 M02)
N260 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
N99999999 %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
N99999999 %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

%UPGREP 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
N99999999 %UPGREP G71 *	

Program execution

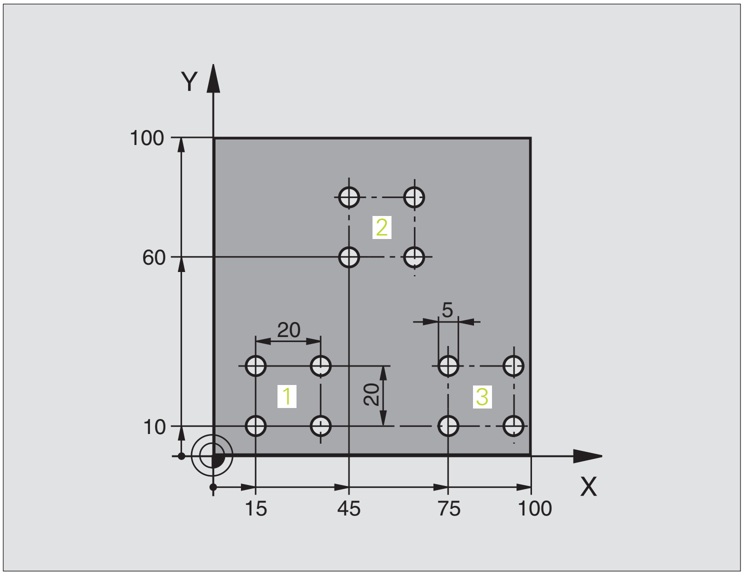
1. Main program UPGREP is executed up to block N110.
2. Subprogram 2 is called and executed.
3. Program section between block N120 and block N100 is repeated twice. Subprogram 2 is repeated twice.
4. Main program UPGREP is executed once from block N130 to block N190. End of program.

10.6 Programming Examples

Example: Milling a contour in several infeeds

Program sequence

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



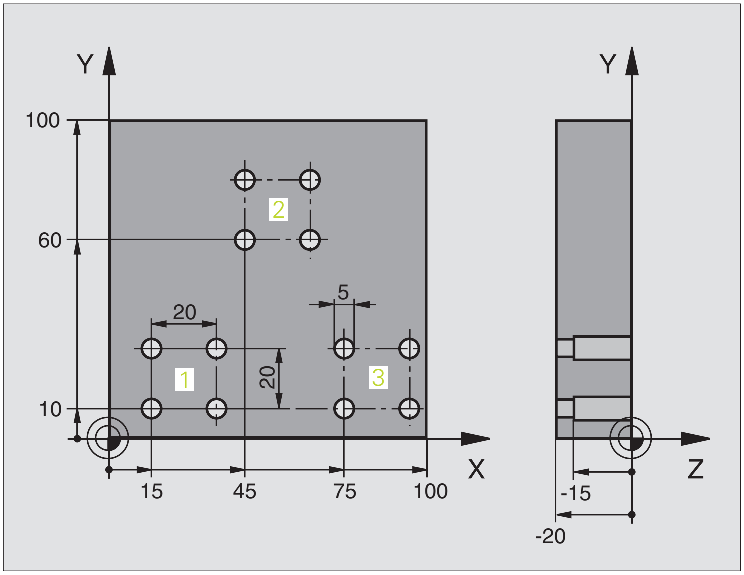
%PGMWDH G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+7.5 *	Define the tool
N40 T1 G17 S3500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 I+50 J+50 *	Set pole
N70 G10 R+60 H+180 *	Pre-position in the working plane
N80 G01 Z+0 F1000 M3 *	Pre-position to 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 to the contour.
N130 H+120 *	
N140 H+60 *	
N150 H+0 *	
N160 H-60 *	
N170 H-120 *	
N180 H+180 *	
N190 G27 R5 F500 *	Depart the contour
N200 G40 R+60 H+180 F1000 *	Retract tool
N210 L1.4 *	Return jump to label 1; section is repeated 4 times
N220 G00 Z+250 M2 *	Retract in the tool axis, end program
N99999999 %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



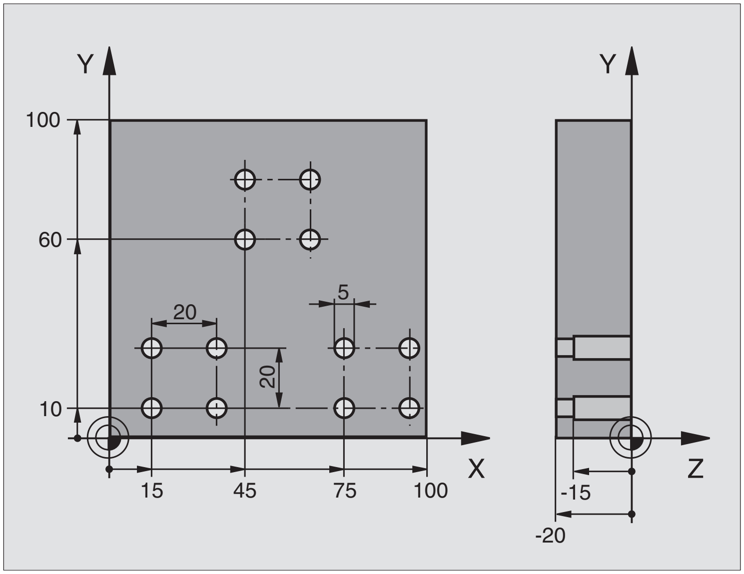
%UP1 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+2.5 *	Define the tool
N40 T1 G17 S3500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=300 ;FEED RATE FOR PLNGNG	
Q202=5 ;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
N99999999 %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

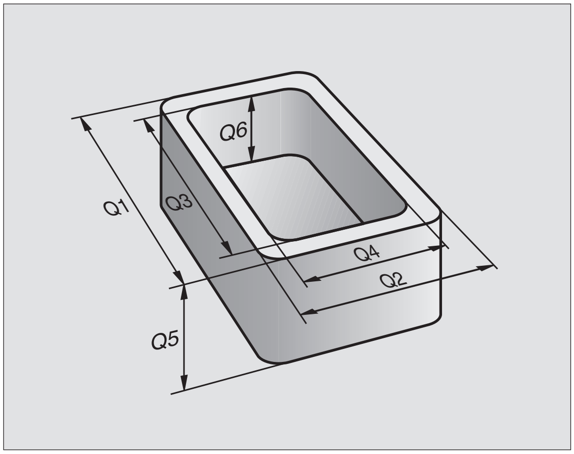
N100 G00 Z+250 M6 *	Tool change
N110 T2 G17 S4000 *	Call tool: drill
N120 D0 Q201 P01 -25 *	New depth for drilling
N130 D0 Q202 P01 +5 *	New plunging depth for drilling
N140 L1.0 *	Call subprogram 1 for the entire hole pattern
N150 G00 Z+250 M6 *	Tool change
N160 T3 G17 S500 *	Call tool: reamer
N80 G201 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 %UP2 G71 *	

11

Programming: Q Parameters

11.1 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 1999. They are grouped according to various ranges:

Meaning	Range
Freely applicable parameters, globally effective for all programs stored in the TNC memory	Q1600 to Q1999
Freely applicable parameters, as long as no overlapping with SL cycles can occur, globally effective for 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 stored in the TNC memory	Q200 to Q1199
Parameters that are primarily used for OEM cycles, globally effective for all programs stored in the TNC memory. This may require coordination with the machine manufacturer or supplier.	Q1200 to Q1399
Parameters that are primarily used for **call-active** OEM cycles, globally effective for all programs that are stored in the TNC memory	Q1400 to Q1499
Parameters that are primarily used for **Def-active** OEM cycles, globally effective for all programs that are stored in the TNC memory	Q1500 to Q1599

QS parameters (the S stands for string) are also available on the TNC and enable you to process texts. In principle, the same ranges are available for QS parameters as for Q parameters (see table above).

Note that for the QS parameters the **QS100** to **QS199** range is reserved for internal texts.

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 536).

If you are using the parameters Q60 to Q99 in encoded 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	Page
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHM.	Page 510
Trigonometric functions	TRIGO- NOMETRY	Page 513
If/then conditions, jumps	JUMP	Page 515
Other functions	DIVERSE FUNCTION	Page 518
Entering formulas directly	FORMULA	Page 524
Function for machining complex contours	CONTOUR FORMULA	Page 415
Function for string processing	STRING FORMULA	Page 528

11.2 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 is assigned the value 25
`N250  G00 X +Q10 *`	Corresponds to G00 X +25

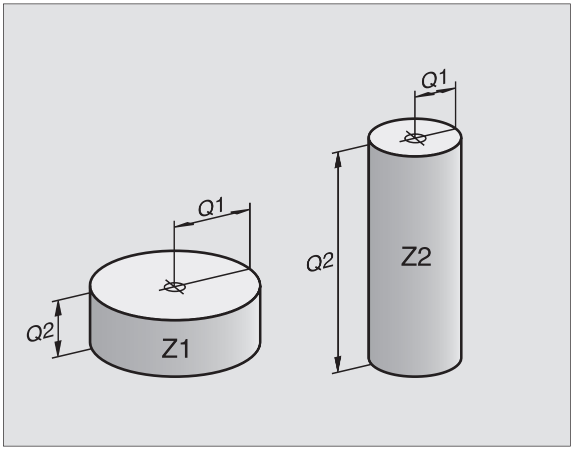
You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

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



11.3 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 *`** Calculates and assigns the difference of two values.	D2 X – Y
D03: MULTIPLICATION Example: **`D03 Q2 P01 +3 P02 +3 *`** Calculates and assigns 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.

1ST 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.

1ST VALUE OR PARAMETER?

Q5 ENT	Enter Q5 for the first value.

2ND VALUE OR PARAMETER?

7 ENT	Enter 7 for the second value.

Example: NC block

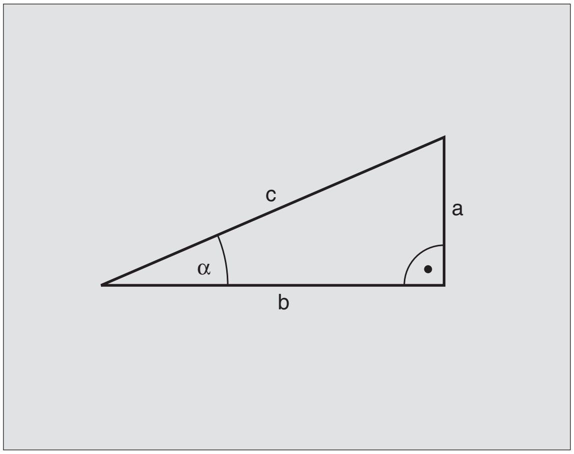
```
N17 D03 Q12 P01 +Q5 P02 +7 *
```

11.4 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

$\alpha = \arctan \alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

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 *`** Calculates the sine of an angle in degrees (°) and assigns 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 *`** Calculates the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assigns it to a parameter.	D13 X ANG Y

11.5 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 490). 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 "UPCAN25" *`** 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 "ANYNAME" *`** 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

11.6 Checking and Editing Q Parameters

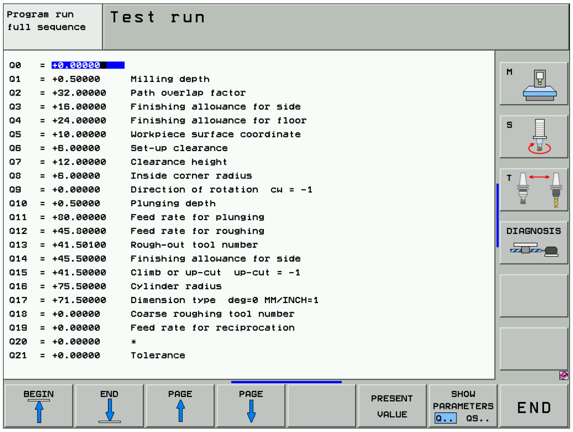
Procedure

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 used by the TNC are provided with comments.

If you want to check or edit string parameters, press the SHOW PARAMETERS Q... QS... soft key. The TNC then displays all string parameters and the above described also apply.

11.7 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	Page
D14:ERROR Output of error messages	D14 ERROR=	Page 519
D15:PRINT Output of unformatted texts or Q parameter values	D15 PRINT	Page 523
FD19:PLC Transfer values to the PLC	D19 PLC=	Page 523

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 message predefined by HEIDENHAIN

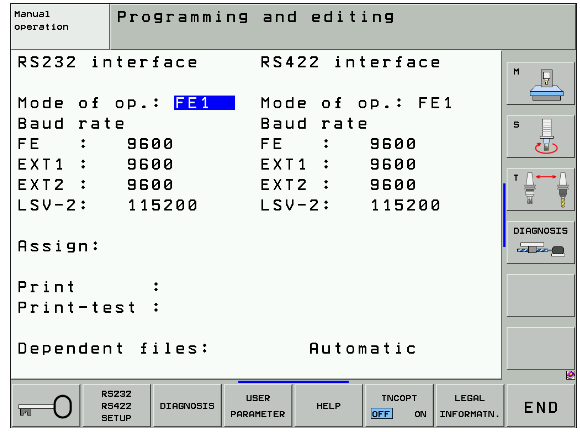
Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING not allowed
1008	MIRRORING not allowed
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Entry value incorrect
1012	Wrong sign programmed
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points

Error number	Text
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

Error number	Text
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTED SPINDLE STOP not allowed
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as a negative value
1078	Q303 not defined in measuring cycle
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory measuring points
1082	Clearance height entered incorrectly
1083	Contradictory type of plunging
1084	Machining cycle not permitted
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not permitted
1090	Enter infeed unequal 0

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 600).

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 100 and above:	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: Transfer 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 *
```

11.8 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, then Q50 >= 0 If result for Q12 = –1, then Q50 < 0	SGN
Calculate modulo value Example: `Q12 = 400 % 360` Result: Q12 = 40	%

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first

```
N112  Q1 = 5 * 3 + 2 * 10 = 35 *
```

1st calculation: 5 * 3 = 15

2nd calculation: 2 * 10 = 20

3rd step 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

```
N30 Q25 = ATAN (Q12/Q13) *
```

11.9 String Parameters

String processing functions

You can use the **QS** parameters to create variable character strings.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) to a string parameter. You can also check and process the assigned or imported values by using the functions described below.

The STRING FORMULA and FORMULA Q-parameter functions contain various functions for processing the string parameters.

STRING FORMULA functions	Soft key	Page
Assigning string parameters	STRING	Page 529
Chain-linking string parameters		Page 529
Converting a numerical value to a string parameter	TOCHAR	Page 530
Copying a substring from a string parameter	SUBSTR	Page 531

FORMULA string functions	Soft key	Page
Converting a string parameter to a numerical value	TONUMB	Page 532
Checking a string parameter	INSTR	Page 533
Finding the length of a string parameter	STRLEN	Page 534
Comparing alphabetic priority	STRCOMP	Page 535

When you use a STRING FORMULA, the result of the arithmetic operation is always a string. When you use the FORMULA function, the result of the arithmetic operation is always a numeric value.

Assigning string parameters

You have to assign a string variable before you use it. Use the DECLARE STRING command to do so.

SPEC FCT
- To select the TNC special functions, press the SPEC FCT key

DECLARE
- Select the DECLARE function

STRING
- Select the STRING soft-key

Example NC block:

```
N37 DECLARE STRING QS10 = "WORKPIECE"
```

Chain-linking string parameters

With the concatenation operator (string parameter ||) you can make a chain of two or more string parameters.

Q
- Select Q parameter functions.

STRING FORMULA
- Select STRING FORMULA function.
- Enter the number of the string parameter in which the TNC is to save the concatenated string. Confirm with the ENT key.
- Enter the number of the string parameter in which the **first** substring is saved. Confirm with the ENT key: The TNC displays the concatenation symbol ||.
- Confirm your entry with the ENT key.
- Enter the number of the string parameter in which the **second** substring is saved. Confirm with the ENT key.
- Repeat the process until you have selected all the required substrings. Conclude with the END key.

Example: QS10 is to include the complete text of QS12, QS13 and QS14

```
N37 QS10 =  QS12 || QS13 || QS14
```

Parameter contents:

- **QS12**: **Workpiece**
- **QS13**: **Status:**
- **QS14**: **Scrap**
- **QS10**: **Workpiece Status: Scrap**

Converting a numerical value to a string parameter

With the **TOCHAR** function, the TNC converts a numerical value to a string parameter. This enables you to chain numerical values with string variables.

Q

- Select Q parameter functions.

STRING FORMULA

- Select STRING FORMULA function.

TOCHAR

- Select the function for converting a numerical value to a string parameter.
- Enter the number or the desired Q parameter to be converted, and confirm with the ENT key.
- If desired, enter the number of decimal places that the TNC should convert, and confirm with the ENT key.
- Close the parenthetical expression with the ENT key and confirm your entry with the END key.

Example: Convert parameter Q50 to string parameter QS11, use 3 decimal places

```
N37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```

Copying a substring from a string parameter

With the **SUBSTR** function you can copy a definable range from a string parameter.

Q

- Select Q parameter functions.

STRING FORMULA

- Select STRING FORMULA function.
- Enter the number of the string parameter in which the TNC is to save the copied string. Confirm with the ENT key.

SUBSTR

- Select the function for cutting out a substring
- Enter the number of the QS parameter from which the substring is to be copied. Confirm with the ENT key.
- Enter the number of the place starting from which to copy the substring, and confirm with the ENT key.
- Enter the number of characters to be copied, and confirm with the ENT key.
- Close the parenthetical expression with the ENT key and confirm your entry with the END key.

Remember that the first character of a text sequence starts internally with the zeroth place.

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2).

```
N37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```

Converting a string parameter to a numerical value

The **TONUMB** function converts a string parameter to a numerical value. The value to be converted should be only numerical.

The QS parameter must contain only one numerical value. Otherwise the TNC will output an error message.

Q

- Select Q parameter functions.

FORMULA

- Select the FORMULA function.
- Enter the number of the string parameter in which the TNC is to save the numerical value. Confirm with the ENT key.

◁

- Shift the soft-key row.

TONUMB

- Select the function for converting a string parameter to a numerical value.
- Enter the number of the Q parameter to be converted, and confirm with the ENT key.
- Close the parenthetical expression with the ENT key and confirm your entry with the END key.

Example: Convert string parameter QS11 to a numerical parameter Q82

```
N37 Q82 = TONUMB ( SRC_QS11 )
```

Checking a string parameter

With the **INSTR** function you can check whether a string parameter is contained in another string parameter.

Q

- Select Q parameter functions.

FORMULA

- Select the FORMULA function.
- Enter the number of the Q parameter in which the TNC is to save the place at which the search text begins. Confirm with the ENT key.

◁

- Shift the soft-key row.

INSTR

- Select the function for checking a string parameter
- Enter the number of the QS parameter in which the searched-for text is saved. Confirm with the ent key.
- Enter the number of the QS parameter to be searched, and confirm with the ENT key.
- Enter the number of the place starting from which the TNC is to search the substring, and confirm with the ENT key.
- Close the parenthetical expression with the ENT key and confirm your entry with the END key.

If the TNC does not find the substring, it saves the value 0 in the result parameter.

If the substring is found in more than one place, the TNC returns the first place at which it finds the substring.

Example: Search through QS10 for the text saved in parameter QS13. Begin the search at the third place.

```
N37 Q50 = INSTR ( SRC_QS10 SEA_QS13 BEG2 )
```

Finding the length of a string parameter

The **STRLEN** function returns the length of the text saved in a selectable string parameter.

Q

- Select Q parameter functions.

FORMULA

- Select the FORMULA function.
- Enter the number of the Q parameter in which the TNC is to save the ascertained string length. Confirm with the ENT key.

◁

- Shift the soft-key row.

STRLEN

- Select the function for finding the text length of a string parameter
- Enter the number of the QS parameter whose length the TNC is to ascertain, and confirm with the ENT key.
- Close the parenthetical expression with the ENT key and confirm your entry with the END key.

Example: Find the length of QS15

```
N37 Q52 = STRLEN ( SRC_QS15 )
```

Comparing alphabetic priority

With the **STRCOMP** function you can compare string parameters for alphabetic priority.

Q
- Select Q parameter functions.

FORMULA
- Select the FORMULA function.
- Enter the number of the Q parameter in which the TNC is to save the result of comparison. Confirm with the ENT key.

◁
- Shift the soft-key row.

STRCOMP
- Select the function for comparing string parameters
- Enter the number of the first QS parameter to be compared, and confirm with the ENT key.
- Enter the number of the second QS parameter to be compared, and confirm with the ENT key.
- Close the parenthetical expression with the ENT key and confirm your entry with the END key.

The TNC returns the following results:

- **0**: The compared QS parameters are identical
- **+1**: The first QS parameter **precedes** the second QS parameter alphabetically.
- **–1**: The first QS parameter **follows** the second QS parameter alphabetically.

Example: QS12 and QS14 are compared for alphabetic priority

```
N37 Q52 = STRCOMP ( SRC_QS12 SEA_QS14 )
```

11.10 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
- Results of measurements from touch probe cycles etc.

Preassigned Q parameter between Q100 and Q199 must not be used in NC programs as calculation parameters in NC programs. Otherwise you might receive undesired results.

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.

WMAT block: QS100

The TNC saves the material defined in the WMAT block in parameter **QS100.**

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

(see also the Touch Probe Cycles User's Manual)

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

Tool measurement with the BLUM laser.	Parameter value
Reserved.	Q190
Reserved.	Q191
Reserved.	Q192
Reserved.	Q193

Reserved for internal use	Parameter value
Markers for cycles (point patterns)	Q197
Number of the active touch probe cycle	Q198

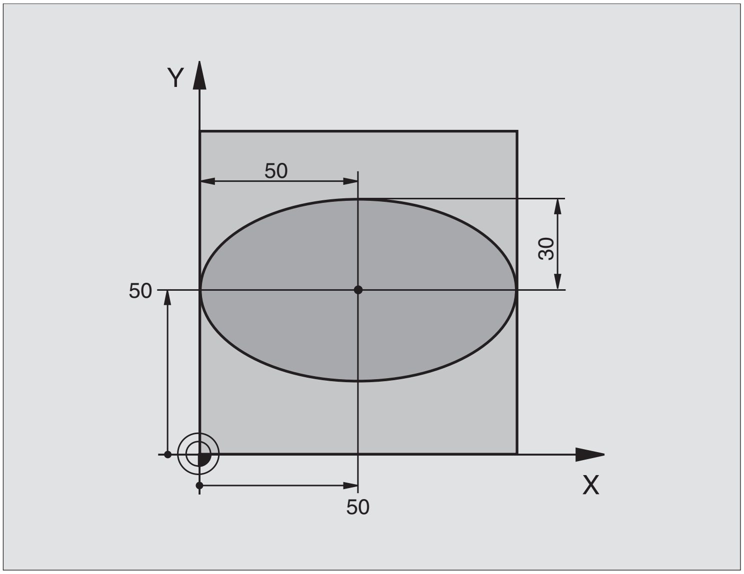
Status during tool measurement with TT	Parameter value
Tool within tolerance	Q199 =0.0
Tool is worn (LTOL/RTOL exceeded)	Q199 =1.0
Tool is broken (LBREAK/RBREAK exceeded)	Q199 =2.0

11.11 Programming Examples

Example: Ellipse

Program sequence

- The contour of the ellipse is approximatedby 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.



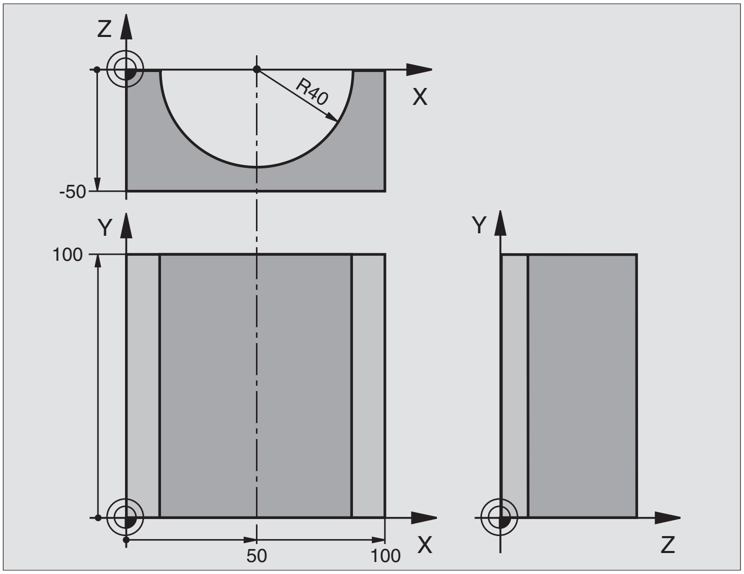
%ELLIPSE G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q3 P01 +50 *	Semiaxis in X
N40 D00 Q4 P01 +30 *	Semiaxis in Y
N50 D00 Q5 P01 +0 *	Starting angle in the plane
N60 D00 Q6 P01 +360 *	End angle in the plane
N70 D00 Q7 P01 +40 *	Number of calculation steps
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
N99999999 %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.



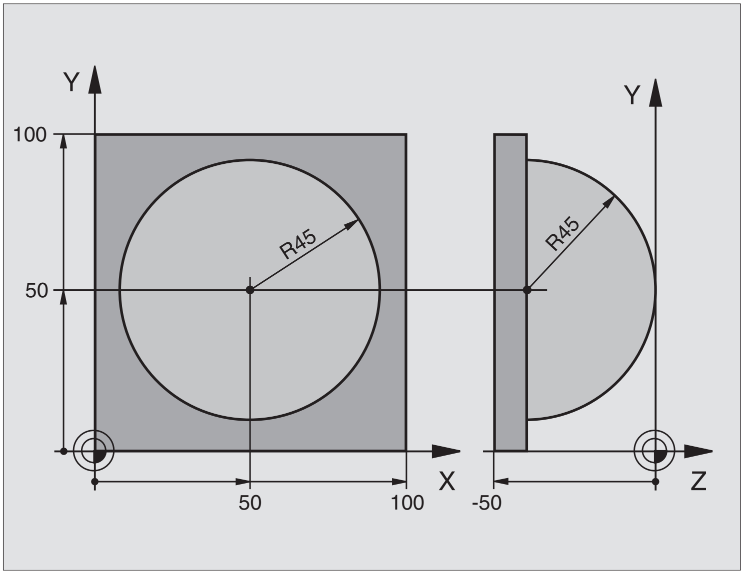
`%CYLIN G71 *`	
`N10 D00 Q1 P01 +50 *`	Center in X axis
`N20 D00 Q2 P01 +0 *`	Center in Y axis
`N30 D00 Q3 P01 +0 *`	Center in Z axis
`N40 D00 Q4 P01 +90 *`	Starting angle in space (Z/X plane)
`N50 D00 Q5 P01 +270 *`	End angle in space (Z/X plane)
`N60 D00 Q6 P01 +40 *`	Radius of the cylinder
`N70 D00 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
`N99999999 %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
N99999999 %SPHERE G71 *	

12

Test Run and Program Run

12.1 Graphics

Function

In the program run modes of operation as well as in the Test Run mode, the TNC provides the following three display modes: Using soft keys, select whether you desire:

- Plan view
- Projection in 3 planes
- 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter. For this purpose, enter R2 = R in the tool table.

The TNC will not show a graphic if

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

With MPs 7315 to 7317 you can have the TNC display a graphic even if no tool axis is defined or moved.

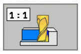
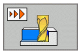
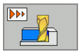
Using the new 3-D graphics you can now also graphically display machining operations in the tilted working plane as well as multi-sided machining operations, provided that you have already simulated the program in another view. The MC 422 B hardware is required to be able to use this function. In order to increase the speed of the test graphics on older hardware versions, bit 5 of MP7310 should be set to 1. This deactivates functions which were implemented specifically for the 3-D graphics.

The TNC graphic does not show a radius oversize DR that has been programmed in the TOOL CALL block.

Setting the speed of the test run

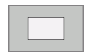
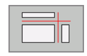
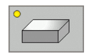
> You can set the speed of the test run only if the "Display of machining time" function is active (see "Activating the stopwatch function" on page 558). The TNC otherwise always performs the test run at the maximum possible speed.
>
> The most recently set speed remains active, even after a restart, until you change it.

After you have started a program, the TNC displays the following soft keys with which you can set the simulation speed.

Function	Soft key
Execute test run at the same speed at which the program will be executed (programmed feed rates are taken into account).	
Increase the test speed incrementally.	
Decrease the test speed incrementally.	
Test run at the maximum possible speed (default setting).	

Overview of display modes

The control displays the following soft keys in the Program Run and Test Run modes of operation:

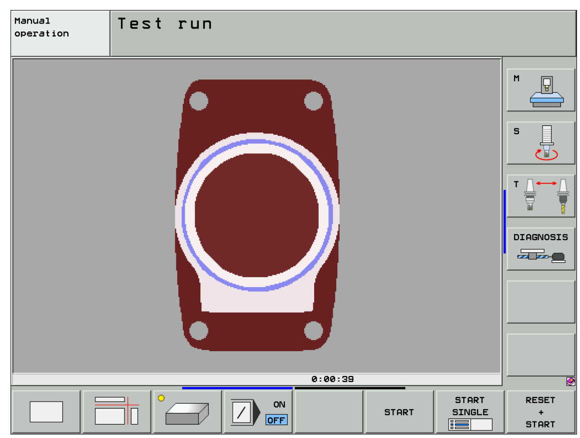
View	Soft key
Plan view	
Projection in 3 planes	
3-D view	

Limitations during program run

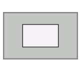
A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text **ERROR** in the graphics window. The machining process is continued, however.

Plan view

If your machine has a mouse, the status bar shows the depth of any location on the workpiece when you move the mouse pointer over it.



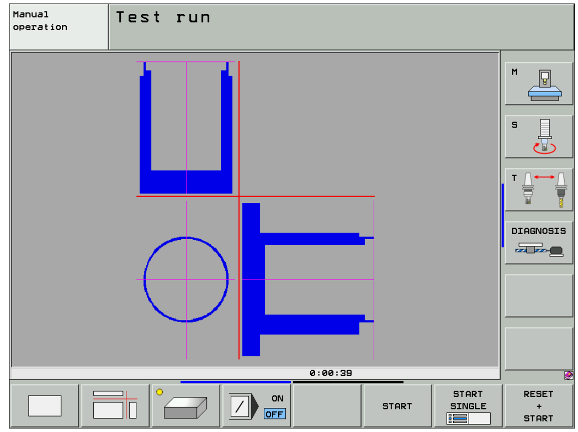
This is the fastest of the three graphic display modes.



- Press the soft key for plan view.
- Regarding depth display, remember:

  The deeper the surface, the darker the shade.

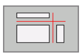
Projection in 3 planes



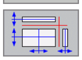
Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

Details can be isolated in this display mode for magnification (see "Magnifying details," page 556).

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

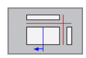
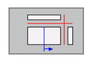
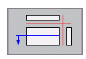
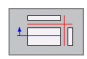
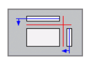
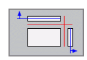


▶ Select the soft key for projection in three planes.



▶ Shift the soft-key row and select the soft key for sectional planes.

▶ The TNC then displays the following soft keys:

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

The positions of the sectional planes are visible during shifting.

The default setting of the sectional plane is selected such that it lies in the working plane in the workpiece center and in the tool axis on the top surface.

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.

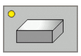
3-D view

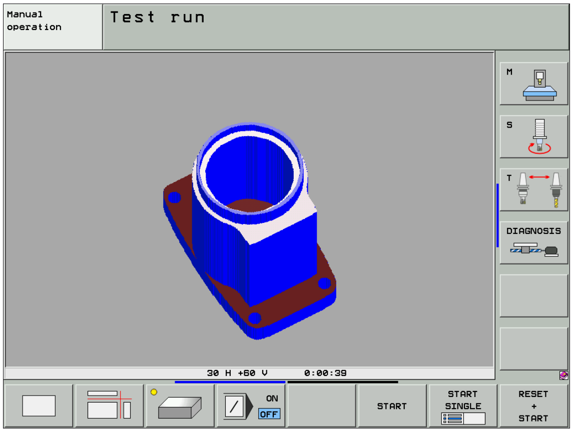
The workpiece is displayed in three dimensions. If you have the appropriate hardware, then with its high-resolution 3-D graphics the TNC can also display machining operations in the tilted working plane as well as multi-sided machining operations.

You can rotate the 3-D display about the vertical and horizontal axes. If there is a mouse attached to your TNC, you can also perform this function by holding down the right mouse button and dragging the mouse.

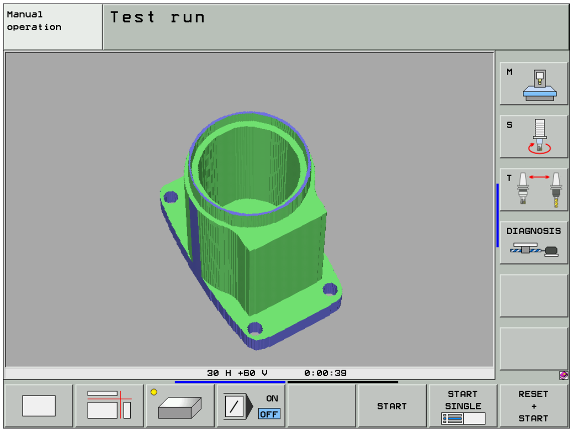
The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation.

In the Test Run mode of operation you can isolate details for magnification, see “Magnifying details,” page 556.





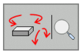
▶ Press the soft key for 3-D view. Press the soft key twice to switch to the high-resolution 3-D graphics. This switch is only possible once the simulation has finished. The high-resolution graphics also display machining operations in the tilted working plane.



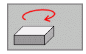
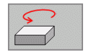
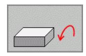
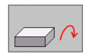
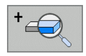
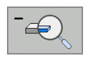
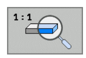
The speed of the high-resolution 3-D graphics depends on the tooth length (LCUTS column in the tool table). If LCUTS is defined as 0 (basic setting), the simulation calculates an infinitely long tooth length, which leads to a long processing time. If you do not want to define LCUTS, then set MP7312 to a value between 5 and 10. This way the TNC internally limits the tooth length to a value which is calculated from MP7312 times the tool diameter.

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:

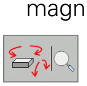
Function	Soft keys
Rotate in 5° steps about the vertical axis	 
Rotate in 5° steps about the horizontal axis	 
Magnify the graphic stepwise. If the view is magnified, the TNC shows the letter **Z** in the footer of the graphic window.	
Reduce the graphic stepwise. If the view is reduced, the TNC shows the letter **Z** in the footer of the graphic window.	
Reset image to programmed size	

If there is a mouse attached to your TNC, you can also perform the functions described above with the mouse.

- In order to rotate the graphic shown in three dimensions: Hold the right mouse button down and move the mouse. In the high resolution 3-D graphics the TNC displays the coordinate system showing the currently active alignment of the workpiece. In the normal 3-D view the entire workpiece rotates as well. After you release the right mouse button, the TNC orients the workpiece to the defined orientation.
- In order to shift the graphic shown: Hold the center mouse button or the wheel button down and move the mouse. The TNC shifts the workpiece in the corresponding direction. After you release the center mouse button, the TNC shifts the workpiece to the defined position.
- In order to zoom in on a certain area with the mouse: Draw a rectangular zoom area while holding the left mouse button down. After you release the left mouse button, the TNC zooms in on the defined area of the workpiece.
- In order to quickly zoom in and out with the mouse: Rotate the wheel button forward or backward.

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:

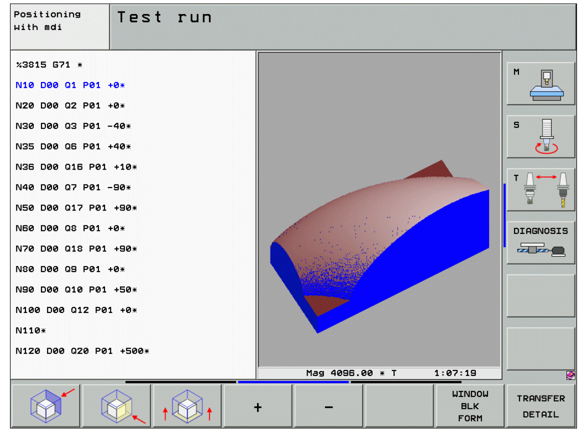
BLK FORM DISPLAY HIDE

- Show the frame for the BLK FORM: Set the highlight in the soft key to SHOW

BLK FORM DISPLAY HIDE

- Hide the frame for the BLK FORM: Set the highlight in the soft key to OMIT

Magnifying details



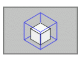
You can magnify details in all display modes in the Test Run mode and a program run mode.

The graphic simulation or the program run, respectively, must first have been stopped. A detail magnification is always effective in all display modes.

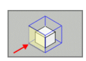
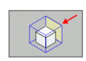
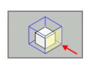
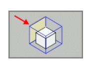
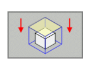
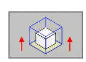
Changing the detail magnification

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 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 it was programmed with BLK FORM.	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.

Displaying the tool

You can display the tool during simulation in the plan view and in the projection in 3 planes. The TNC depicts the tool in the diameter defined in the tool table.

Function	Soft key
Do not display the tool during simulation	TOOLS DISPLAY HIDE
Display the tool during simulation	TOOLS DISPLAY HIDE

Measuring the machining time

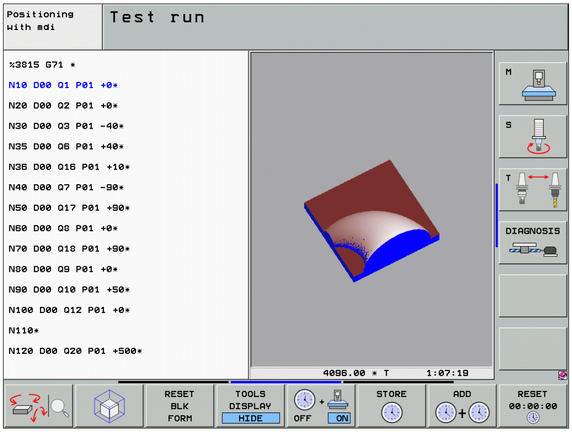
Program Run modes of operation

The timer counts and displays the time from program start to program end. The timer stops whenever machining is interrupted.

Test Run

The timer displays the time that the TNC calculates from the duration of tool movements, including dwell times calculated by the TNC. The time calculated by the TNC can only conditionally be used for calculating the production time because the TNC does not account for the duration of machine-dependent interruptions, such as tool change.

If you have switched the "find machining time" function on, you can generate a file listing the usage times of all tools used in the program (see “Dependent files” on page 612).



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
Enable (ON) or disable (OFF) the "measure the machining time" function.	OFF ON
Store displayed time	STORE
Display the sum of stored time and displayed time	ADD
Clear displayed time	RESET 00:00:00

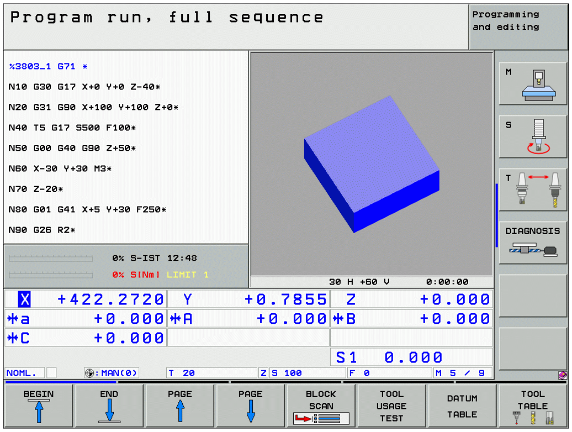
The soft keys available to the left of the stopwatch functions depend on the selected screen layout.

During the Test Run, the TNC resets the machining time as soon as a new **BLK FORM** is evaluated.

12.2 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 ↓

12.3 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

Some traverse motions performed by the machine cannot be graphically simulated by the TNC. These include

- traverse motions during tool change, if the machine manufacturer defined them in a tool-change macro or via the PLC,
- positioning movements that the machine manufacturer defined in an M-function macro,
- positioning movements that the machine manufacturer performs via the PLC, and
- positioning movements that lead to a pallet change.

HEIDENHAIN therefore recommends proceeding with caution for every new program, even when the program test did not output any error message, and no visible damage to the workpiece occurred.

After a tool call, the TNC always starts a program test at the following position:

- In the working plane, at the **MIN** point given in the workpiece blank definition
- In the tool axis, 1 mm above the **MAX** point defined in the workpiece blank definition

If you call the same tool, the TNC resumes program simulation from the position last programmed before the tool call.

In order to ensure unambiguous behavior during program run, after a tool change you should always move to a position from which the TNC can position the tool for machining without causing a collision.

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 614).

- Select the Test Run operating mode
- Call the file manager with the PGM MGT key and select the file you wish to test, or
- Go to the program beginning: Select line “0” with the GOTO key and confirm your entry with the ENT key.

The TNC then displays the following soft keys:

Function	Soft key
Reset the blank form and test the entire program	RESET + START
Test the entire program	START
Test each program block individually	START SINGLE
Halt program test (soft key only appears once you have started the program test)	STOP

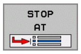
You can interrupt the program test and continue it again at any point—even within a machining cycle. In order to continue the test, the following actions must not be performed:

- Selecting another block with the GOTO key
- Making changes to the program
- Switching the operating mode
- Selecting a new program

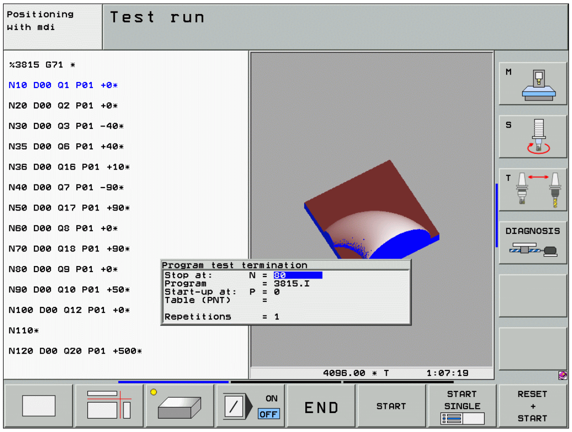
Running a program test up to a certain block

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

- Go to the beginning of program in the Test Run mode of operation.
- To 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 PGM CALL, 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.



12.4 Program Run

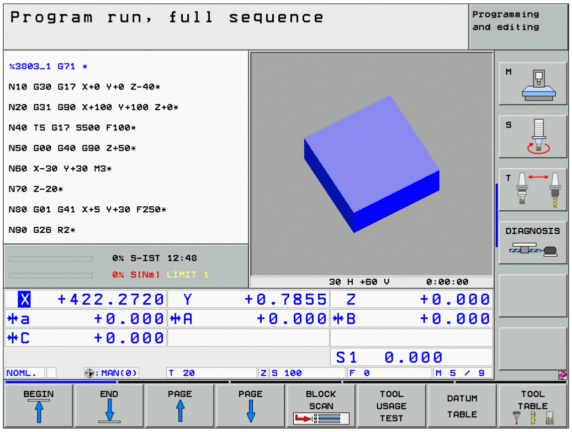
Function

In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or up to a program stop.

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

The following TNC functions are available in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Optional block skip
- Editing the tool table TOOL.T
- Check and change Q parameters
- Superimpose handwheel positioning
- Functions for graphic simulation
- Additional status display



Run a part program

Preparation

1. Clamp the workpiece to the machine table.
2. Set the datum.
3. Select the necessary tables and pallet files (status M).
4. Select the part program (status M).

> You can adjust the feed rate and spindle speed with the override knobs.
>
> It is possible to reduce the rapid traverse speed when starting the NC program using the FMAX soft key. The reduction applies to all rapid traverse and feed rate movements. The value you enter is no longer in effect after the machine has been turned off and on again. In order to re-establish the respectively defined maximum feed rate after switch-on, you need to re-enter the corresponding value.

Program Run, Full Sequence

- Start the part program with the machine START button.

Program Run, Single Block

- Start each block of the part program individually with the machine START button.

Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- Pressing the machine STOP button
- 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** (with and without a miscellaneous function)
- Miscellaneous functions **M0**, **M2** or **M30**
- Miscellaneous function **M6** (defined by the machine tool builder)

Interruption through 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.

Programming of noncontrolled axes (counter axes)

> This function must be adapted by your machine manufacturer. Refer to your machine manual.

The TNC automatically interrupts the program run as soon as an axis is programmed in a positioning block that was defined by the machine tool builder as a noncontrolled axis (counter axis). In this condition you can move the non-controlled axis manually to the desired position. In the left window, the TNC shows all nominal positions programmed in this block. In noncontrolled axes, the TNC also displays the distance remaining.

As soon as all axes are in the correct position, you can use NC Start to resume program run.

RESTORE X

- Select the desired axis sequence and start each with NC Start. Manually position the non-controlled axes. The TNC shows the distance remaining to the nominal position in this axis (see “Returning to the contour” on page 570).

OFF ON

- If required, choose whether the close-loop axes are to be moved in the tilted or non-tilted coordinate system.

MANUAL TRAVERSE

- If required, move the axes by handwheel or with the axis-direction buttons.

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 switch the coordinate system between tilted and non-tilted, as well as to the active tool axis direction, by pressing the 3-D ROT 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 TRAVERSE 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.

Your machine tool builder can define whether in a program interruption you always move the axes in the currently active (tilted or non-tilted) coordinate system. 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.

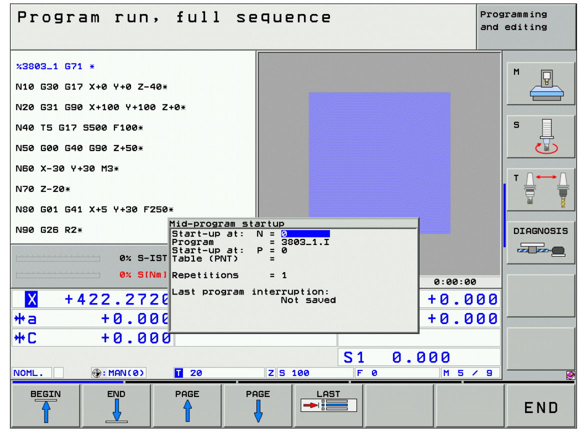
With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.

If the program was interrupted by one of the conditions described below, the TNC saves the point of interruption.

- EMERGENCY STOP
- Power interruption
- Control software crash

After you have called the mid-program startup function, you can press the soft key SELECT LAST N to reactivate the point of interruption and approach it with an NC start. After switch-on the TNC then shows the message **`NC program was interrupted.`**



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 applies if you have only changed the 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.

With the 3-D ROT soft key, you can switch the coordinate system between tilted and non-tilted in order to move to the start-up position.

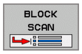
If you want to use the block scan feature in a pallet table, select the program in which a mid-program startup is to be performed from the pallet table by using the arrow keys. Then press the RESTORE POS AT N soft key.

The TNC skips all touch probe cycles in a mid-program startup. Result parameters that are written to from these cycles might therefore remain empty.

The **M142/M143** functions are not allowed during a mid-program startup.

If you execute mid-program startup in a program containing M128, the TNC performs compensating movements as necessary. The compensating movements are superposed over the approach movement.

- To go to the first block of the current program to start a block scan, enter GOTO "0".

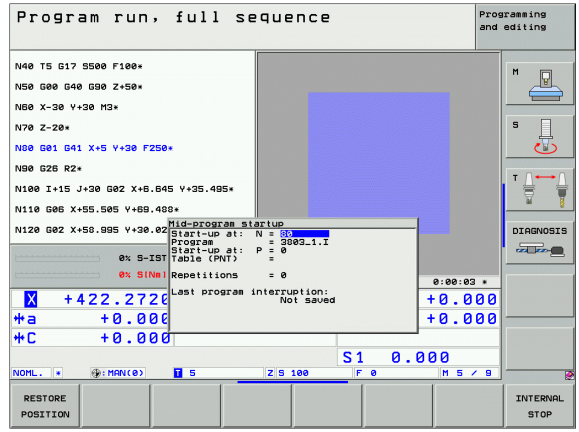


- To select block scan, press the BLOCK SCAN soft key.
- **Start-up at N:** Enter the block number N at which the block scan should end.
- **Program:** Enter the name of the program containing block N.
- **Repetitions:** If block N is located in a program section repeat, enter the number of repetitions to be calculated in the block scan.
- To start the block scan, press the machine START button.
- Contour approach (see following section).

Returning to the contour

With the RESTORE POSITION function, the TNC returns to the workpiece contour in the following situations:

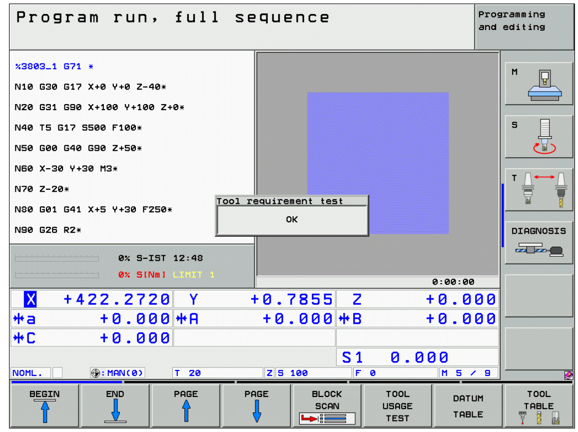
- Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function.
- Return to the contour after a block scan with RESTORE POS AT N, for example after an interruption with INTERNAL STOP.
- Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption
- If a non-controlled axis is also programmed in a positioning block (see "Programming of noncontrolled axes (counter axes)" on page 565)



- ▶ To select a return to contour, press the RESTORE POSITION soft key.
- ▶ Restore machine status, if required.
- ▶ To move the axes in the sequence that the TNC suggests on the screen, press the machine START button.
- ▶ To move the axes in any sequence, press the soft keys RESTORE X, RESTORE Z, etc., and activate each axis with the machine START key.
- ▶ To resume machining, press the machine START key.

Tool usage test

The tool usage test function must be enabled by your machine manufacturer. Refer to your machine manual.



The following are prerequisites for a tool usage test:

- Bit 2 of the machine parameter must be set to 7246=1
- the machining timer must be active in the **Test Run** operating mode
- A simulation of the plain language program must have been completed in the **Test Run** mode

With the TOOL USAGE TEST soft key, you can check before starting a program in a Program Run operating mode whether the tool being used has enough service life remaining. Here the TNC compares the actual values for service life in the tool table with the nominal values from the tool requirement file.

After you have clicked the soft key, the TNC displays the results of the tool usage test in a pop-up window. Use the CE key to close the pop-up window.

The TNC saves the usage times in a separate file with the extension **pgmname.H.T.DEP**. (see “Changing the MOD setting for dependent files” on page 612). The generated tool usage file has the following information:

Column	Meaning
TOKEN	■ **TOOL:** Tool usage time per **TOOL CALL**. The entries are listed in chronological order. ■ **TTOTAL:** Total usage time of a tool ■ **STOTAL:** Call of a subprogram (including cycles). The entries are listed in chronological order. ■ **TIMETOTAL:** The total machining time of the NC program is entered in the **WTIME** column. In the **PATH** column the TNC saves the path name of the corresponding NC programs. The **TIME** column shows the sum of all **TIME** entries (only when the spindle is on, and without rapid traverse). The TNC sets all other columns to 0 ■ **TOOLFILE:** In the **PATH** column, the TNC saves the path name of the tool table with which you conducted the Test Run. This enables the TNC during the actual tool usage test to detect whether you did with test run with the TOOL.T.
TNR	Tool number (**–1**: No tool inserted yet)
IDX	Tool index
NAME	Tool name from the tool table

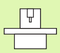
Column	Meaning
TIME	Tool usage time in seconds
RAD	**Tool radius R** + **Oversize of tool radius DR** from the tool table. The unit is 0.1 µm
BLOCK	Block number in which the **TOOL CALL** block was programmed
PATH	■ **TOKEN** = **TOOL:** Path name of the active main program or subprogram ■ **TOKEN** = **STOTAL:** Path name of the subprogram

There are two ways to run a tool usage test for a pallet file:

- The highlight is on a pallet entry in the pallet file:
  The TNC runs the tool usage test for the entire pallet.
- The highlight is on a program entry in the pallet file:
  The TNC runs the tool usage test for the selected program.

12.5 Automatic Program Start

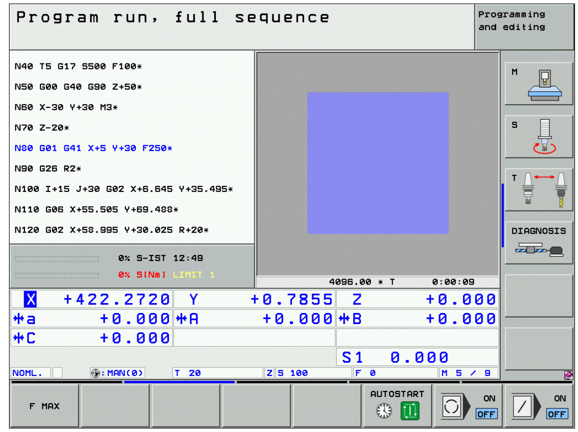
Function

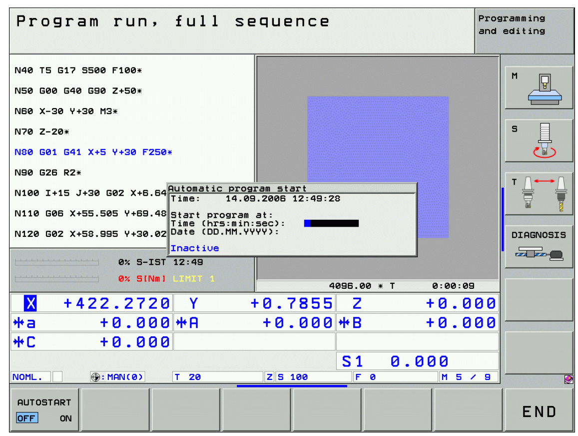


The TNC must be specially prepared by the machine tool builder for use of the automatic program start function. Refer to your machine manual.

In a Program Run operating mode, you can use the AUTOSTART soft key (see figure at upper right) to define a specific time at which the program that is currently active in this operating mode is to be started:

- 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.





12.6 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.

12.7 Optional Program-Run Interruption

Function

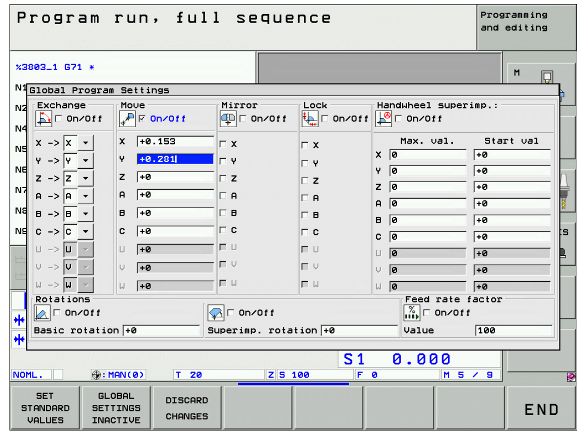
The TNC optionally interrupts the program run or test run at blocks containing M01. If you use M01 in the Program Run mode, the TNC does not switch off the spindle or coolant.

- Do not interrupt Program Run or Test Run at blocks containing M01: Set soft key to OFF.

- Interrupt Program Run or Test Run at blocks containing M01: Set soft key to ON.

12.8 Global Program Settings (Software Option)

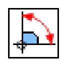
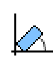
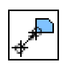
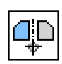
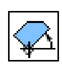
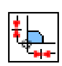
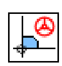
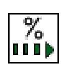
Function



The **global program settings,** which are used in particular for large molds and dies, are available in the Program Run mode and MDI mode. You can use it to define various coordinate transformations and settings that are globally effective and are superimposed on the respectively selected NC program so that you do not need to edit the NC program.

You can activate and deactivate global program settings, even in mid-program if you have interrupted the program run (see "Interrupting machining" on page 564).

The following global program settings are available:

Function	Icon	Page
Exchanging axes		Page 579
Basic rotation		Page 579
Additional, additive datum shift		Page 580
Superimposed mirroring		Page 580
Superimposed rotation		Page 581
Axis locking		Page 581
Definition of a handwheel superimposition		Page 582
Definition of a globally effective feed rate factor		Page 581

You cannot use global program run settings if you have used the **M91/M92** function (moving to machine-referenced positions) in your NC program.

You can use the look-ahead function **M120** if you have activated the global program settings before starting the program. If **M120** is active and you change global settings during the program, the TNC will show an error message and stop any further machining.

If dynamic collision monitoring (DCM) is active, you cannot define a handwheel superimposition.

In the fillable form the TNC grays out any axes that are not active on your machine.

Activating/deactivating a function

Global program settings remain active until you manually reset them.

If a global program setting is active, the TNC shows the symbol in the position display.

If you use the file management to select a program, the TNC displays a warning message if global settings are active. Then you can simply acknowledge the message with the soft key or call the form directly to make the changes.

Global program settings have no effect in the smarT.NC operating mode.

- Select the Program Run or Manual Data Input operating mode.
- Shift the soft-key row.
- Call the global program settings form
- Activate the desired functions with the corresponding values

If you activate more than one global program setting, the TNC calculates the transformations internally in the following sequence:

- **1**: Axis exchange
- **2**: Basic rotation
- **3**: Shift
- **4**: Mirror image
- **5**: Superimposed rotation

The remaining functions such as axis locking, handwheel superimposition and feed rate factor act independently.

The functions in the following list help you to navigate in the form. You can also use the mouse to use the form.

Function	Key/ Soft key
Jump to previous function	
Jump to next function	
Select the next element	
Select the previous element	
Axis exchange function: Open the list of available axes	GOTO
Switch the function on/off if the cursor is on a checkbox.	SPACE
Reset the global program settings: ■ Deactivate all functions ■ Set all entered value to 0, set feed rate factor to 100. Set basic rotation to 0 if no preset is active in the preset table. Otherwise the TNC defines the preset at the basic rotation entered in the preset table.	SET STANDARD VALUES
Discard all changes since the form was last called	DISCARD CHANGES
Deactivate all active functions. The entered or adjusted values remain.	GLOBAL SETTINGS INACTIVE
Save all changes and close the form	END

Exchanging axes

With the axis exchanging function you can adapt the axes programmed in any NC program to your machine's axis configuration or to the respective clamping situation.

> After activation of the axis exchange function, all subsequent transformations are applied to the exchanged axes.
>
> Be sure to exchange the axes appropriately. Otherwise the TNC will display error messages.
>
> Remember that may have to return to the contour after activation of this function. The TNC then automatically calls the return-to-contour menu after the form is closed (see "Returning to the contour" on page 570).

- In the global program settings form, move the cursor to **EXCHANGE ON/OFF,** and use the SPACE key to activate the function.
- With the downward arrow key, set the cursor to the line showing at left the axis to be exchanged.
- Press the Goto key to display the list of axes with which you can exchange it.
- With the downward arrow key, select the axes with which you wish to exchange, and confirm with the ENT key.

If you work with a mouse, you can select the desired axis directly by clicking it in the respective pull-down menu.

Basic rotation

The basic rotation function enables you to compensate a workpiece misalignment. The effect corresponds to the basic rotation function that you can define in the manual mode with the probing functions. The TNC therefore saves the new value simultaneously in the form and in the basic rotation menu, although only one is visible.

> Remember that may have to return to the contour after activation of this function. The TNC then automatically calls the return-to-contour menu after the form is closed (see "Returning to the contour" on page 570).

Additional, additive datum shift

With the additive datum shift function you can compensate any offsets in all active axes.

> The values defined in the form work in addition to the values already defined in the program through Cycle **G53** or **G54** (datum shift).
>
> Remember that may have to return to the contour after activation of this function. The TNC then automatically calls the return-to-contour menu after the form is closed (see “Returning to the contour” on page 570).

Superimposed mirroring

With the superimposed mirroring function you can mirror all active axes.

> The mirrored axes defined in the form work in addition to the values already defined in the program through Cycle 8 (mirroring).
>
> Remember that may have to return to the contour after activation of this function. The TNC then automatically calls the return-to-contour menu after the form is closed (see “Returning to the contour” on page 570).

- In the global program settings form, move the cursor to **MIRRORING ON/OFF,** and use the SPACE key to activate the function.
- With the downward arrow key, set the cursor to the axis that you want to mirror.
- Press the SPACE key to mirror the axis. Pressing the SPACE key again cancels the function.

If you work with a mouse, you can select the desired axis directly by clicking it.

Superimposed rotation

With the superimposed rotation function you can define any rotation of the coordinate system in the presently active working plane.

> The superimposed rotation defined in the form works in addition to the values already defined in the program through Cycle **G73** (rotation).
>
> Remember that may have to return to the contour after activation of this function. The TNC then automatically calls the return-to-contour menu after the form is closed (see “Returning to the contour” on page 570).

Axis locking

With this function you can lock all active axes. Then when you run a program, the TNC does not move any of the axes you locked.

> When you activate this function, ensure that the positions of the locked axes cannot cause any collisions.

- In the global program settings form, move the cursor to **LOCK ON/OFF,** and activate the function with the SPACE key.
- With the downward arrow key, set the cursor to the axis that you want to lock.
- Press the SPACE key to lock the axis. Pressing the SPACE key again cancels the function.

If you work with a mouse, you can select the desired axis directly by clicking it.

Feed rate factor

With the feed rate factor function, you can decrease or increase the programmed feed rate by a percentage. The input range is 1% to 1000%.

> Remember that the TNC always applies the feed rate factor to the current feed rate, which you may already have changed through the feed rate override.

Handwheel superimposition

The handwheel superimposition function enables you to use the handwheel to move the axes while the TNC is running a program.

In the **Max. val.** column you define the maximum distance by which you can move the axis by handwheel. As soon as you interrupt the program run (control-in-operation signal is off), the TNC shows the distances actually moved in each axis in the **Start val** column. The start value remains saved until you delete it, even after a power interruption. You can also edit the **start value.** If required, the TNC decreases the value that you entered to the respective **Max. val.**

If a **start value** is shown during activation, then when the window closes, the TNC calls the "Return to contour" function to move by the defined value (see "Returning to the contour" on page 570).

A maximum traverse distance, defined in the NC programmed with **M118,** is overwritten by the value entered in the form. In turn, the TNC enters distances that have already been traversed with the handwheel using **M118** in the **start value** column of the form so that there is no jump in the display during activation. If the distance already traversed with **M118** is greater than the maximum permissible value in the form, then when the window closes the TNC calls the "return to contour" function in order to move by the difference value (see "Returning to the contour" on page 570).

If you try to enter a **start value** greater than the **max. value,** the TNC shows an error message. Never enter a **start value** greater that the **Max. value.**

12.9 Adaptive Feed Control Software Option (AFC)

Function

The **AFC** feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.

Adaptive feed control is not intended for tools with diameters less than 5 mm. This limit diameter might also be greater if the spindle's rated power is very high.

Do not work with adaptive feed control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.

In adaptive feed control the TNC automatically controls the feed rate during program run as a function of the current spindle power consumption. The spindle power required for each machining step is to be recorded in a teach-in cut and saved by the TNC in a file belonging to the part program. When each machining step is started, which is normally when the spindle is switched on with **M3,** the TNC controls the feed rate so that it remains within the limits that you have defined.

This makes it possible to avoid negative effects on the tool, the workpiece, and the machine that might be caused by changing cutting conditions. Cutting conditions are changed particularly by:

- Tool wear
- Fluctuating cutting depths that occur especially with cast parts
- Fluctuating hardness caused by material flaws

Adaptive feed control (AFC) offers the following benefits:

- Optimization of machining time
  By controlling the feed rate, the TNC tries to maintain the recorded maximum spindle power during the entire machining time. It shortens the machining time by increasing the feed rate in machining zones with little material removal.
- Tool monitoring
  If the spindle power exceeds the recorded maximum value, the TNC decreases the feed rate until the reference spindle power is reattained. If the maximum spindle power is exceeded during machining and at the same time the feed rate falls below the minimum that you defined, the TNC reacts by shutting down. This helps to prevent further damage after a tool breaks or is worn out.
- Protection of the machine's mechanical elements
  Timely feed rate reduction and shutdown responses help to avoid machine overload.

Defining the AFC basic settings

You make the control settings for the TNC feed rate control in the table **AFC.TAB,** which must be saved in the root directory **TNC:\.**

The data in this table are default values that were copied during a teach-in cut into a file belonging to the respective program and serve as the basis for control. The following data are to be defined in this table:

Column	Function
NR	Consecutive line number in the table (has no further functions)
AFC	Name of the control setting. You enter this name in the **AFC** column of the tool table. It specifies the assignment of control parameters to the tool.
FMIN	Feed rate at which the TNC is to conduct a shutdown response. Enter the value in percent with respect to the programmed feed rate. Input range: 50 to 100%
FMAX	Maximum feed rate in the material up to which the TNC can automatically increase the feed rate. Enter the value is in percent with respect to the programmed feed rate.
FIDL	Feed rate for traverse when the tool is not cutting (feed rate in the air). Enter the value is in percent with respect to the programmed feed rate.
FENT	Feed rate for traverse when the tool moves into or out of the material. Enter the value in percent with respect to the programmed feed rate. Maximum input value: 100%
OVLD	Desired reaction of the TNC to overload: ■ **M**: Execution of a macro defined by the machine tool builder ■ **S**: Immediate NC stop ■ **F**: NC stop if the tool has been retracted ■ **E**: Just display an error message on the screen ■ **-**: No overload reaction The TNC conducts a shutdown response if the maximum spindle power is exceeded for more than one second and at the same time the feed rate falls below the minimum you defined.
POUT	Spindle power at which the TNC is to detect when the tool exits the workpiece. Enter the value in percent with respect to the recorded reference load. Recommended value: 8%

Column	Function
SENS	Sensitivity (aggressivity) of the control. Input possible between 50 and 200. 50 is for slow control, 200 for a very aggressive control. An aggressive control reacts quickly and with strong changes to the values, but it tends to overshoot. Recommended value: 100
PLC	Value that the TNC is to transfer to the PLC at the beginning of a machining step. The machine tool builder defines the function, so refer to your machine manual.

In the **AFC.TAB** table you can define as many control settings (lines) as desired.

If there is no AFC.TAB table in the **TNC:** directory, the TNC uses permanently defined internal control settings for the teach-in cut. It is best, however, to work with the AFC.TAB table.

Proceed as follows to create the AFC.TAB file (required only if the file does not yet exist):

- Select the **Programming and Editing** mode of operation.
- To call the file manager, press the PGM MGT soft key.
- Select the **TNC:** directory.
- Make the new file **AFC.TAB** and confirm with the ENT key: The TNC shows a list of table formats.
- Select the **AFC.TAB** table format and confirm with the ENT key: The TNC creates a table with the **Standard** control settings.

Recording a teach-in cut

In a teach-in cut, first the TNC copies for each machining step the basic settings defined in the AFC.TAB table into the **<name>.I.AFC.DEP** file. **<Name>** is for the name of the NC program for which you have recorded the teach-in cut. In addition, the TNC measures the maximum spindle power consumed during the teach-in cut and saves this value in the table.

Each line in the **<name>.I.AFC.DEP** file stands for a machining step that you start with **M3** (or. **M4**) and end with **M5.** You can edit all data of the **<name>.I.AFC.DEP** file if you wish to optimize them. If you have optimized the values in comparison with the values in the AFC.TAB table, the TNC places an asterisk * in front of the control settings in the AFC column. Besides the data from the AFC.TAB table (see “Defining the AFC basic settings” on page 585), the TNC saves the following additional information in the **<name>.I.AFC.DEP** file:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was made (not editable)
IDX	Index of the tool with which the machining step was made (not editable)
N	Difference for tool call: ■ **0**: Tool was called by its tool number. ■ **1**: Tool was called by its tool name.
PREF	Reference load of the spindle. The TNC measures the value in percent with respect to the rated power of the spindle.
ST	Status of the machining step ■ **L**: In the next program run, a teach-in cut is recorded for this machining step. The TNC overwrites any existing values in this line. ■ **C**: The teach-in cut was successfully completed. The next program run can be conducted with automatic feed control.
AFC	Name of the control setting

Remember the following before you record a teach-in cut:

- If required, adapt the control settings in the AFC.TAB table
- Enter the desired control setting for all tools in the **AFC** column of the tool table TOOL.T.
- Select the program for teach-in.
- Activate the adaptive feed control by soft key (see "Activating/ deactivating AFC" on page 590).

When you record a teach-in cut, the TNC internally sets the spindle override to 100%. Then you can no longer change the spindle speed.

You need not run the complete machining step in the teach-in mode. If you can no longer significantly change the cutting conditions, then you can immediately switch to the servo control mode. Press the EXIT LEARNING soft key, and the status changes from **L** to **C.**

During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.

You can repeat a teach-in cut as often as desired. Manually reset the **ST** status to **L.** It may be necessary to repeat the teach-in cut if the programmed feed rate was far too fast and forces you to sharply decrease the feed rate override during the machining step.

For a tool, you can teach-in as many machining steps as desired. A machining step always begins with **M3** (or **M4**) and ends with **M5.**

The TNC changes the status from teach-in **(L)** to controlling **(C)** only when the recorded reference load is greater than 2%. Adaptive feed control is not possible for smaller values.

You machine tool builder can provide a feature with which the teach-in cut can be automatically ended after a selectable time. The machine tool manual provides further information.

Proceed as follows to select and, if required, edit the **<name>.I.AFC.DEP** file:

- Select the **Program Run, Full Sequence** operating mode.
- Shift the soft-key row.

AFC SETTINGS

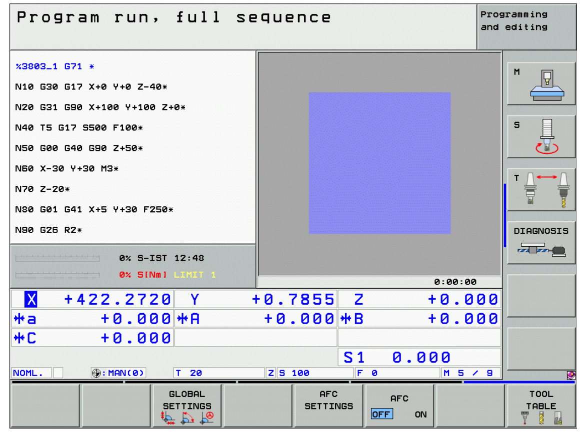
- Select the table of AFC settings.
- Make optimizations if required

Note that the **<name>.I.AFC.DEP** file is locked to editing as long as the NC program **<name>.H** is running. The TNC then displays the data in the table in red.

The TNC removes the editing lock if one of the following functions has been executed:

- M02
- M30
- END PGM

Activating/deactivating AFC



- Select the **Program Run, Full Sequence** operating mode.
- Shift the soft-key row.
- To activate the adaptive feed control: Set the soft key to ON, and the TNC displays the AFC symbol in the position display (see "Status Displays" on page 51).
- To deactivate the adaptive feed control: Set the soft key to OFF.

The adaptive feed control remains active until you deactivate it by soft key.

If the adaptive feed control is active in the **control** mode, the TNC internally sets the spindle override to 100%. Then you can no longer change the spindle speed.

If the adaptive feed control is active in the **control** mode, the TNC takes over the feed rate override function:

- If you increase the feed rate override, it has no influence on the control.
- If you decrease the feed rate override by more than **10%** with respect to the maximum setting, the TNC switches the adaptive feed control off. In this case the TNC displays a window to inform you.

The adaptive feed control is **not active** in NC blocks containing **G0.**

Mid-program startup is allowed during active feed rate control and the TNC takes the cut number of the startup point into account.

In the additional status display, the TNC displays various information when the adaptive feed control is active (see "Adaptive Feed Control (AFC tab, software option)" on page 59). In addition, the TNC shows the symbol in the position display.

Log file

In a teach-in cut, the TNC saves for each machining step relevant data in the **<name>.I.AFC2.DEP** file. **<Name>** is for the name of the NC program for which you have recorded the teach-in cut. During control, the TNC updates the data and makes various evaluations. The following data are to be saved in this table:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was made
IDX	Index of the tool with which the machining step was made
SNOM	Nominal spindle speed [rpm]
SDIF	Maximum difference of the spindle speed in % from the nominal speed
LTIME	Machining time for the teach-in cut
CTIME	Machining time for the control cut
TDIFF	Time difference in % between the machining time during teach-in and control
PMAX	Maximum recorded spindle power during machining. The TNC displays the value in percent with respect to the rated power of the spindle.
PREF	Reference load of the spindle. The TNC displays the value in percent with respect to the rated power of the spindle.
OVLD	Reaction made by the TNC to overload: ■ **M**: A macro defined by the machine tool builder has been run ■ **S**: Immediate NC stop was conducted ■ **F**: NC stop was conducted after the tool was been retracted ■ **E**: An error message was displayed ■ **-**: There was no overload reaction
BLOCK	Block number at which the machining step begins

The TNC records the total machining time for all teach-in cuts **(LTIME)**, all control cuts **(CTIME)** and the total time difference **(TDIFF)**, and enters it after the keyword **TOTAL** in the last line of the log file.

Proceed as follows to select the **<name>.I.AFC2.DEP** file:

- Select the **Program Run, Full Sequence** operating mode.
- Shift the soft-key row.
- AFC SETTINGS: Select the table of AFC settings.
- TABLE EVALU-ATION: Show the log file.

13

MOD Functions

13.1 MOD Functions

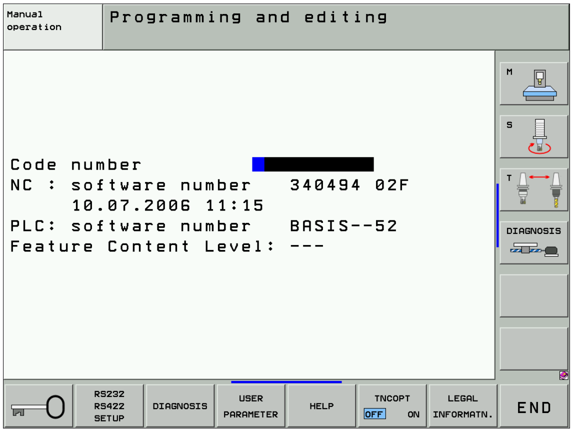
The MOD functions provide additional input possibilities and displays. The available MOD functions depend on the selected operating mode.

Selecting the MOD functions

Call the operating mode in which you wish to change the MOD functions.

MOD

- To select the MOD functions, press the MOD key. The figures at right show typical screen menus in Programming and Editing (figure at upper right), Test Run (figure at lower right) and in a machine operating mode (see figure on next page).



Changing the settings

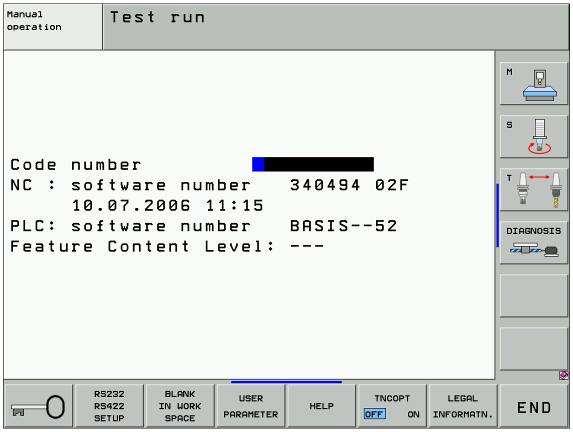
- Select the desired MOD function in the displayed menu with the arrow keys.

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.

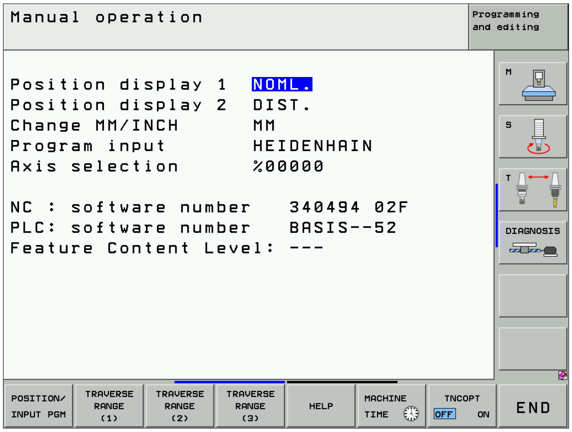
Exiting the MOD functions

- Close the MOD functions with the END key or END soft key.



Overview of MOD functions

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



Programming and Editing:

- Display software numbers
- Enter code number
- Set data interface
- Machine-specific user parameters (if provided)
- Display HELP files (if provided)
- Load service packs
- Time zone, setting
- Legal information

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)
- Time zone, setting
- Legal information

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
- Axis traverse limits
- Display reference points
- Display operating time
- Display HELP files (if provided)
- Time zone, setting
- Legal information

13.2 Software 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)
- **Feature Content Level (FCL):** Development level of the software installed on the control (see “Feature content level (upgrade functions)” on page 8)
- **DSP1** to **DSP3:** Number of the speed controller software (managed by HEIDENHAIN)
- **ICTL1** and **ICTL3:** Number of the current controller software (managed by HEIDENHAIN)

13.3 Entering Code Numbers

Function

The TNC requires a code number for the following functions:

Function	Code number
Select user parameters	123
Configure an Ethernet card (not iTNC 530 with Windows 2000)	NET123
Enable special functions for Q-parameter programming	555343

In addition, you can use the keyword **version** to create a file containing all current software numbers of your control:

- Enter the keyword **version** and confirm with the ENT key.
- The TNC displays all current software numbers on the screen.
- To terminate the version overview, press the END key.

If necessary, you can output the file **version.a** saved in the directory TNC:, and send it to your machine manufacturer or HEIDENHAIN for diagnostic purposes.

13.4 Loading Service Packs

Function

We strongly recommend contacting your machine tool builder before you install a service pack.

The TNC restarts the system after the installation procedure is completed. Before loading the service pack, put the machine in the EMERGENCY STOP condition.

Connect the network drive from which you want to import the service pack (if not already done).

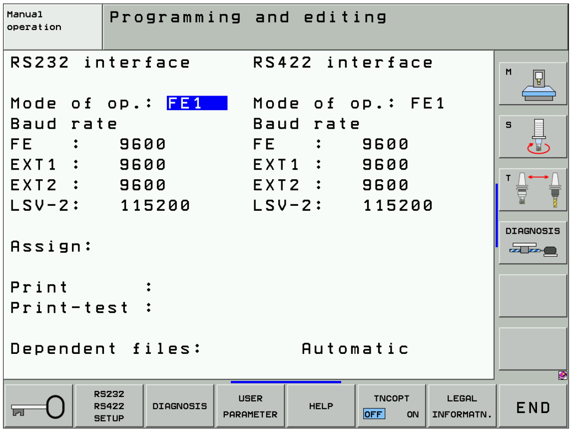
This function provides a simple way of updating the software of your TNC.

- Select the **Programming and Editing** mode of operation.
- Press the MOD key.
- To start the software update, press the "Load Service Pack" soft key. The TNC then displays a superimposed window for selecting the update file.
- Use the arrow keys to select the directory in which the service pack is stored. The respective subdirectories can be shown by pressing the ENT key.
- To select the file: Press the ENT key twice on the selected directory. The TNC switches from the directory window to the file window.
- To start the updating process, press the ENT key to select the file. The TNC unpacks all required files and then restarts the control. This process may take several minutes.

13.5 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 FN15
- Transferring values with Q parameter function FN16

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 FN15/FN16.	- vacant -

File names

Data	Operating mode	File name
Values with FN15	Program Run	%FN15RUN.A
Values with FN15	Test Run	%FN15SIM.A
Values with FN16	Program Run	%FN16RUN.A
Values with FN16	Test Run	%FN16SIM.A

Software for data transfer

For transfer of files to and from the TNC, we recommend using the HEIDENHAIN TNCremoNT data transfer software. With TNCremoNT, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.

> You can download the current version of TNCremoNT free of charge from the HEIDENHAIN Filebase (www.heidenhain.de, <service>, <download area>, <TNCremo NT>).

System requirements for TNCremoNT:

- PC with 486 processor or higher
- Operating system Windows 95, Windows 98, Windows NT 4.0, Windows 2000
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

Installation under Windows

- Start the SETUP.EXE installation program with the File Manager (Explorer).
- Follow the setup program instructions.

Starting TNCremoNT under Windows

- Click <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremoNT>

When you start TNCremoNT for the first time, TNCremoNT automatically tries to set up a connection with the TNC.

Data transfer between the TNC and TNCremoNT

Check whether the TNC is connected to the correct serial port on your PC or to the network, respectively.

Once you have started TNCremoNT, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using the menu items <File> and <Change directory>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- Select <File>, <Setup connection>. TNCremoNT now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**.
- To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click, and drag and drop the highlighted file into the PC window **1**.
- To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**.

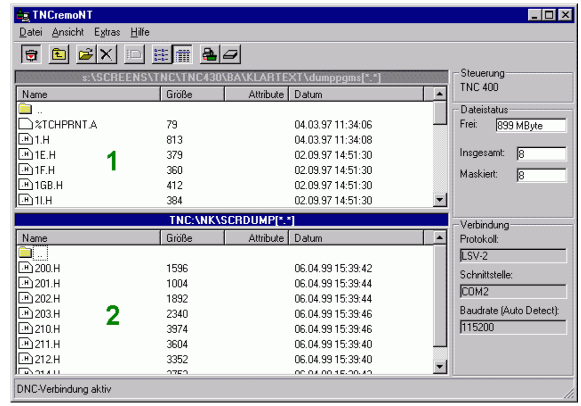
If you want to control data transfer from the TNC, establish the connection with your PC in the following manner:

- 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 PGM MGT key (see "Data transfer to or from an external data medium" on page 123) and transfer the desired files.

End TNCremoNT

Select the menu items <File>, <Exit>.

Refer also to the TNCremoNT context-sensitive help texts where all of the functions are explained in more detail. The help texts must be called with the F1 key.



13.6 Ethernet Interface

Introduction

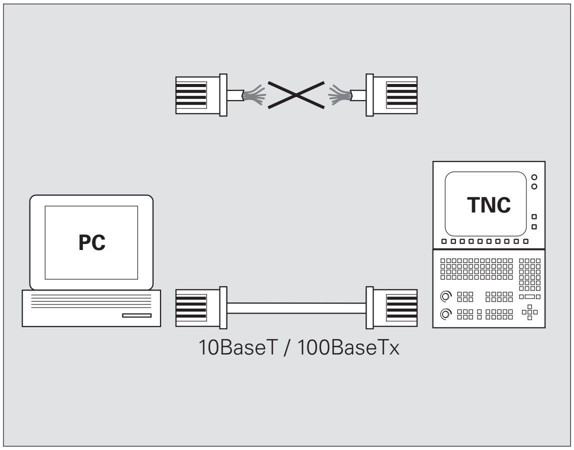
The TNC is shipped with a standard Ethernet card to connect the control as a client in your network. The TNC transmits data via the Ethernet card with

- the **smb** protocol (**Server Message Block**) for Windows operating systems, or
- the **TCP/IP** protocol family (Transmission Control Protocol/Internet Protocol) and with support from the NFS (Network File System). The TNC also supports the NFS V3 protocol, which permits higher data transfer rates.

Connection possibilities

You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 100BaseTX or 10BaseT), or directly to a PC. The connection is metallically isolated from the control electronics.

For a 100BaseTX or 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.

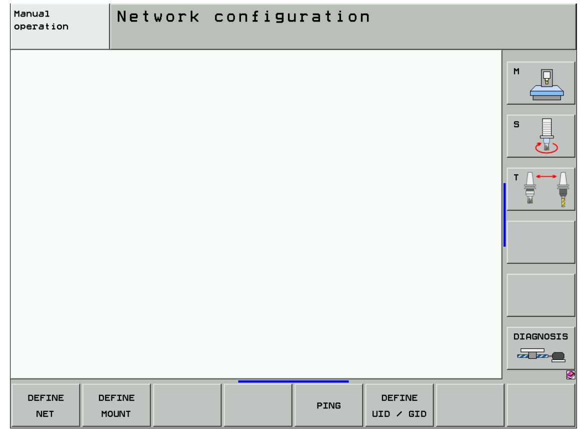


The maximum cable length between TNC and a node depends on the quality grade of the cable, the sheathing and the type of network (100BaseTX or 10BaseT).

If you connect the TNC directly with a PC you must use a transposed cable.

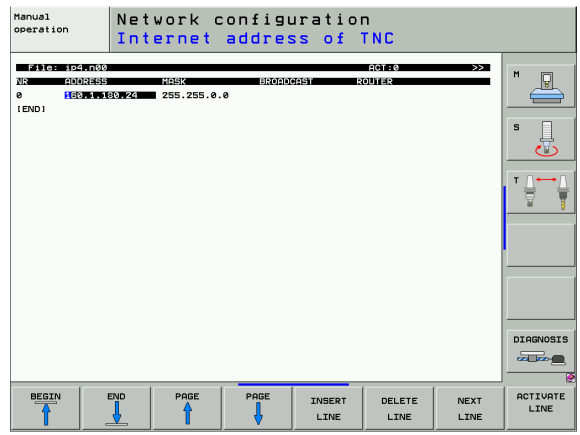
Connecting the iTNC directly with a Windows PC

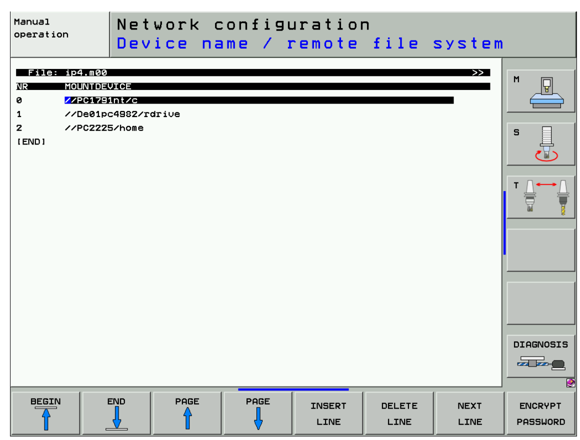
You don’t need any large effort or special networking knowledge to attach the iTNC 530 directly to a PC that has an Ethernet card. You simply have to make some settings on the TNC and the corresponding settings on the PC.



Settings on the iTNC

- Connect the iTNC (connection X26) and the PC with a crossed Ethernet cable (trade names: crossed patch cable or STP cable).
- In the Programming and Editing mode of operation, press the MOD key. Enter the keyword NET123. The iTNC will then display the main screen for network configuration (see figure at top right).
- Press the DEFINE NET soft key to enter the network setting for a specific device (see figure at center right).
- Enter any network address. Network addresses consist of four numbers separated by periods, e.g. **`160.1.180.23`**
- Press the right arrow key to select the next column, and enter the subnet mask. The subnet mask also consists of four numbers separated by periods, e.g. **`255.255.0.0`**
- Press the END key to leave the network configuration screen.
- Press the DEFINE MOUNT soft key to enter the network settings for a specific PC (see figure at bottom right).
- Define the PC name and drive that you want to access, beginning with two slashes, e.g. **`//PC3444/C`**.
- Press the right arrow key to select the next column, and enter the name that the iTNC’s file manager uses to display the PC, e.g. **`PC3444:`**
- Press the right arrow key to select the next column, and enter the file system type **`smb`**
- Press the right arrow key to select the next column and enter the following information (depending on the PC operating system): **`ip=160.1.180.1,username=abcd,workgroup=SALES,password=uvwx`**
- To exit the network configuration, press the END key twice. The iTNC restarts automatically.





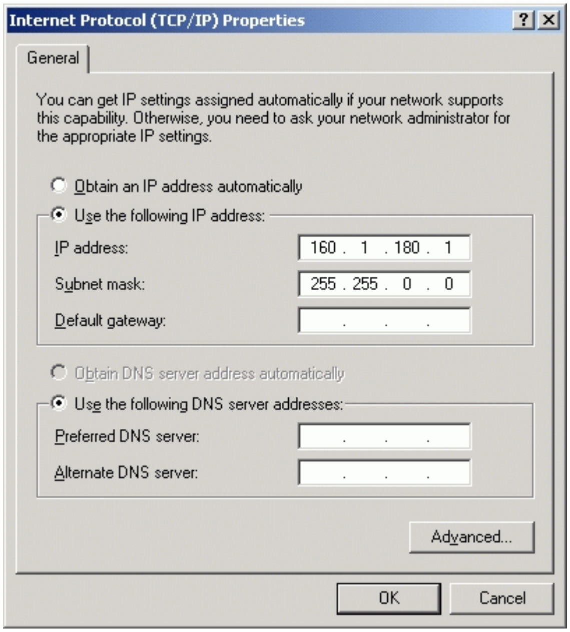
> The parameters **`username,`** **`workgroup`** and **`password`** do not need to be entered in all Windows operating systems.

Settings on a PC with Windows 2000

Prerequisite:

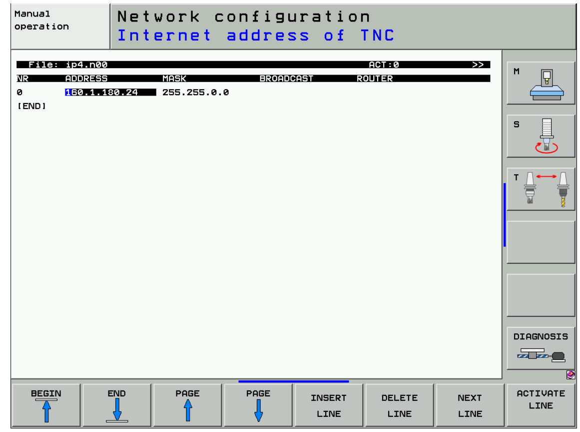
The network card must already be installed on the PC and ready for operation.

If the PC that you want to connect the iTNC to is already integrated in your company network, then keep the PC's network address and adapt the iTNC's network address accordingly.



- To open Network Connections, click <Start>, <Control Panel>, <Network and Dial-up Connections>, and then Network Connections.
- Right-click the <LAN connection> symbol, and then <Properties >in the menu that appears.
- Double-click <Internet Protocol (TCP/IP)> to change the IP settings (see figure at top right).
- If it is not yet active, select the <Use the following IP address> option.
- In the <IP address> input field, enter the same IP address that you entered for the PC network settings on the iTNC, e.g. 160.1.180.1
- Enter 255.255.0.0 in the <Subnet mask> input field.
- Confirm the settings with <OK>.
- Save the network configuration with <OK>. You may have to restart Windows now.

Configuring the TNC



> To configure the dual-processor version: See “Network Settings,” page 661.
>
> Make sure that the person configuring your TNC is a network specialist.
>
> Please note that the TNC performs an automatic reset if you change the IP address of the TNC.

- 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:

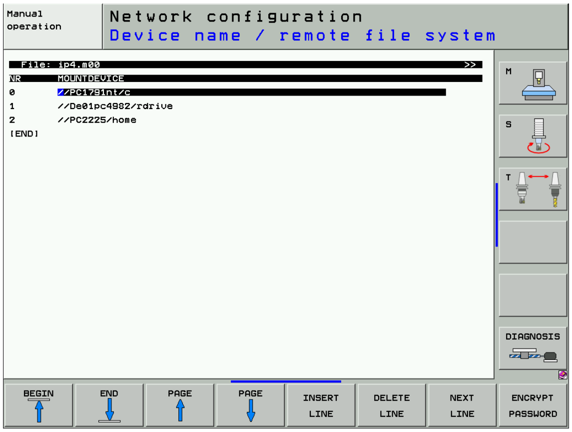
Setting	Meaning
ADDRESS	Address that your network specialist must assign to the TNC. Input: four numerical values separated by points, e.g. 160.1.180.20 As an alternative, the TNC can dynamically retrieve the IP address from a DHCP server. In this case, enter **DHCP**. Note: The DHCP connection is an FCL 2 function.
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	Name of a domain in your company network

Setting	Meaning
NAMESERVER	Network address of the domain server. If DOMAIN and NAMESERVER are defined, you can use symbolic PC names in the mount table, obviating the need for entering the IP address. As an alternative, you can also assign DHCP for dynamic management.

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
MOUNT-DEVICE	■ 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 via smb: Enter the network name and the share name of the computer, e.g. //PC1791NT/C
MOUNT--POINT	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`: **Server Message Block** (Windows protocol)

Setting	Meaning
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 **TIMEO=**: 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.
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 with which the TNC is to log on (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.

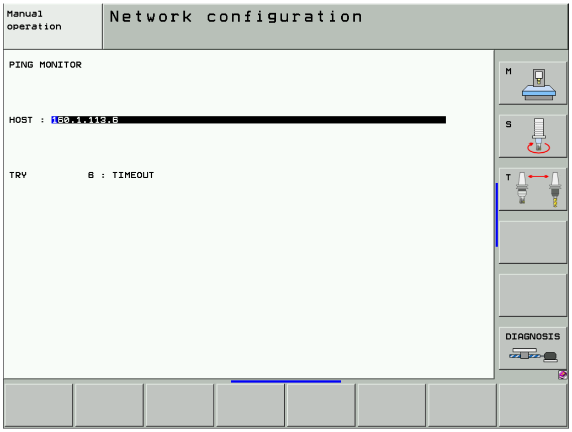
With the ENCODE PASSWORD soft key, you can encode the password defined under OPTIONS.

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.

Test network connection



- Press the PING soft key.
- In the **HOST** line, enter then Internet address of the computer for which you want to check the network connection.
- Confirm your entry with the ENT key. The TNC transmits data packets until you exit the test monitor by pressing the END key.

In the **TRY** line the TNC shows the number of data packets that were transmitted to the previously defined addressee. Behind the number of transmitted data packets the TNC shows the status:

Status display	Meaning
HOST RESPOND	Data packet was received again, connection is OK.
TIMEOUT	Data packet was not received, check the connection.
CAN NOT ROUTE	Data packet could not be transmitted. Check the Internet address of the server and of the router to the TNC.

13.7 Configuring PGM MGT

Function

Use the MOD functions to specify which directories or files are to be displayed by the TNC:

- **PGM MGT** setting: Simple file management (directories are not displayed) or enhanced file management (directories are displayed).
- **Dependent files** setting: Specify whether dependent files are displayed.

Please note: See "Working with the File Manager," page 111.

Changing the PGM MGT setting

- To 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.
- To 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.**

Dependent files

In addition to the file extension, dependent files also have the extension **.SEC.DEP** (**SEC**tion, **DEP**endent). The following different types are available:

- **.I.SEC.DEP**
  The TNC creates files with the **.SEC.DEP** extension if you work with the structure function. The file contains information needed by the TNC to rapidly jump from one structure point to the next.
- **.T.DEP:** Tool usage file for individual conversational-format programs(see "Tool usage test" on page 571)
- **.P.T.DEP:** Tool usage file for a complete pallet
  The TNC creates files with the **.P.T.DEP** ending if, in a Program Run mode, you run the tool usage test (see "Tool usage test" on page 571) for a pallet entry of the active pallet file. This file then lists the sum of all tool usage times of all tools that you use within a pallet.
- **.I.AFC.DEP:** File in which the TNC saves the control parameters for the adaptive feed control (AFC) (see "Adaptive Feed Control Software Option (AFC)" on page 583).
- **.I.AFC2.DEP:** File in which the TNC saves the static data of the adaptive feed control (AFC) (see "Adaptive Feed Control Software Option (AFC)" on page 583).

Changing the MOD setting for dependent files

- To 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.
- To select the Dependent files setting: Using the arrow keys, move the highlight onto the **Dependent files** setting and use the ENT key to switch between **AUTOMATIC** and **MANUAL.**

Dependent files are only visible in the file manager if you selected the MANUAL setting.

If dependent files exist for a file, then the TNC displays a **+** character in the status column of the file manager (only if **Dependent files** is set to **AUTOMATIC**).

13.8 Machine-Specific User Parameters

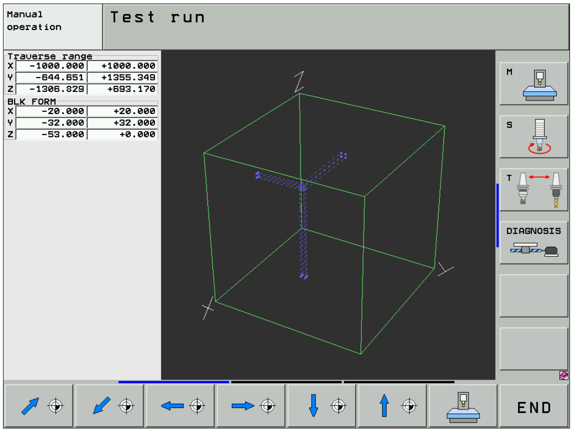
Function

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.

13.9 Showing the Workpiece in the Working Space

Function



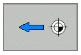
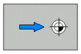
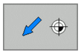
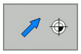
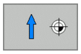
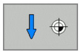
This MOD function enables you to graphically check the position of the workpiece blank in the machine's working space and to activate work space monitoring in the Test Run mode of operation.

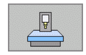
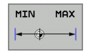
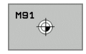
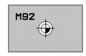
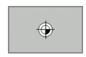
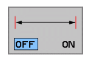
The TNC displays a transparent cuboid for the working space. Its dimensions are shown in the **Traverse range** table (standard color is green). The TNC takes the dimensions for the working space from the machine parameters for the active traverse range. Since the traverse range is defined in the reference 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 transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table (standard color is blue). The TNC takes the dimensions from the workpiece blank definition of the selected program. The workpiece cuboid defines the coordinate system for input. Its datum lies within the traverse-range cuboid. You can view the position of the active datum within the traverse range by pressing the "Show tool datum" soft-key (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 following table.

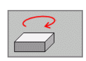
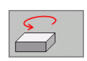
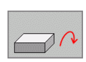
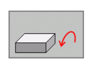
You can also activate the working-space monitor for the Test Run mode in order to test the program with the current datum and the active traverse ranges (see table below, last line).

Function	Soft key
Move workpiece blank to the left	
Move workpiece blank to the right	
Move workpiece blank forward	
Move workpiece blank backward	
Move workpiece blank upward	
Move workpiece blank downward	

Function	Soft key
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	

Rotate the entire image

The third soft-key row provided functions with which you can rotate and tilt the entire image:

Function	Soft keys	
Rotate the image about the vertical axis		
Tilt the image about the horizontal axis		

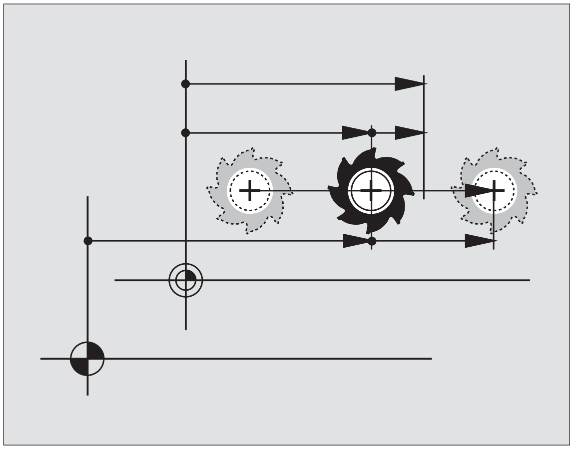
13.10 Position Display Types

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.

13.11 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.

13.12 Selecting 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

13.13 Selecting the Axes for Generating L Blocks

Function

The axis selection input field enables you to define the current tool position coordinates that are transferred to an L block. To generate a separate L block, press the ACTUAL-POSITION-CAPTURE soft key. The axes are selected by bit-oriented definition similar to programming the machine parameters:

Axis selection %11111: Transfer the X, Y, Z, IV, and V axes

Axis selection %01111: X, Y, Z, IV. Transfer the axis

Axis selection %00111: Transfer the X, Y and Z axes

Axis selection %00011: Transfer the X and Y axes

Axis selection %00001: Transfer the X axis

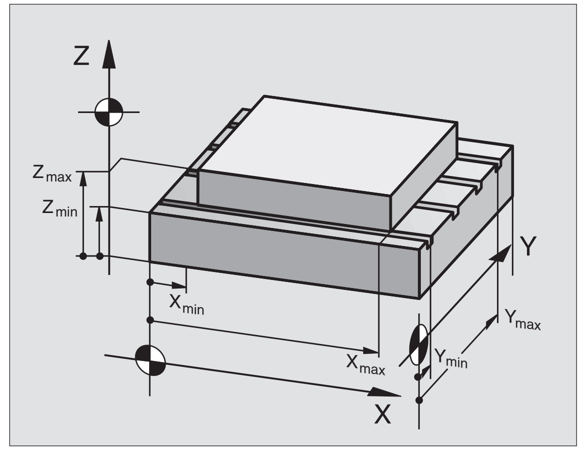
13.14 Entering the Axis Traverse Limits, Datum Display

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

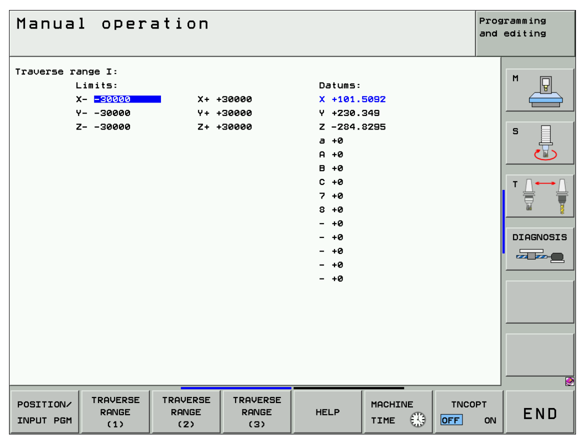
To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 99 999 mm) as the TRAVERSE RANGE.

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.

TRAVERSE RANGE

- Enter the limits for axis traverse: Press the TRAVERSE RANGE soft key and enter the values that you wrote down as limits in the corresponding axes
- To exit the MOD function, press the END soft key.



Active tool radius compensations are not taken into account in the axis traverse limit values.

The traverse range limits and software limit switches become active as soon as the reference points are traversed.

Datum display

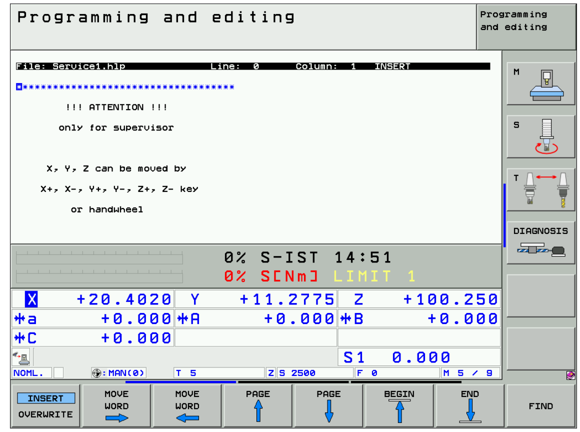
The values shown at the top right of the screen define the currently active datum. The datum can have been set manually or can have been activated from the preset table. The datum cannot be changed in the screen menu.

The displayed values depend on your machine configuration. Refer to the notes in Chapter 2 (see "Explanation of values saved in the preset table" on page 84).

13.15 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.

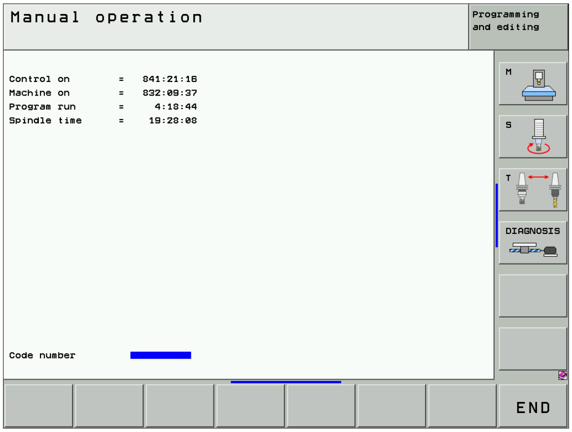
13.16 Displaying Operating Times

Function

> The machine tool builder can provide further operating time displays. The machine tool manual provides further information.

The MACHINE TIME soft key enables you to see various types of operating times:

Operating time	Meaning
Control ON	Operating time of the control since put into service
Machine ON	Operating time of the machine tool since put into service
Program Run	Duration of controlled operation since put into service



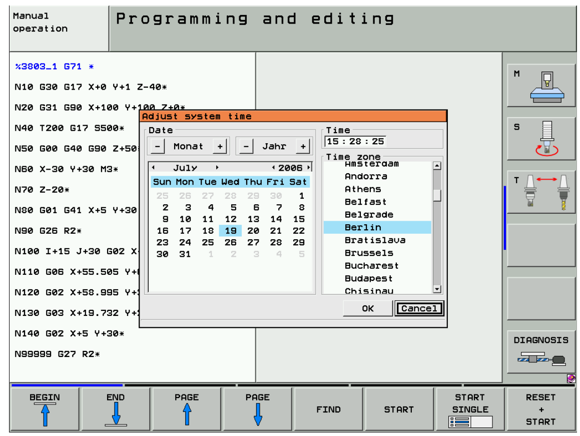
13.17 Setting the System Time

Function

You can set the time zone, the date and the system time with the SET DATE/TIME soft key.

Selecting appropriate settings

> The TNC must be reset after you change the time zone, date or system time. In such cases the TNC displays a warning when the window closes.



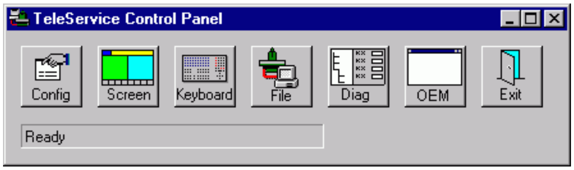
- Press the MOD key to select the MOD function.
- Scroll through the soft-key row.

- To display the time zone window, press the SET TIME ZONE soft key.
- In the left side of the pop-up window, use the mouse to set the year, month and date.
- In the right side under "time zone," click your correct time zone.
- If required, edit the time of day through the keyboard.
- To save the settings, click the **OK** button.
- To discard the changes and cancel the dialog, click the **Cancel** button.

13.18 TeleService

Function

> The TeleService functions are enabled and adapted by the machine tool builder. The machine tool manual provides further information.
>
> The TNC provides two soft keys for teleservice, making it possible to configure two different service agencies.



The TNC allows you to carry out teleservice. To be able to use this feature, your TNC should be equipped with an Ethernet card which achieves a higher data transfer rate than the serial RS232-C interface.

With the HEIDENHAIN TeleService software, your machine tool builder can then establish a connection to the TNC via an ISDN modem and carry out diagnostics. The following functions are available:

- On-line screen transfer
- Polling of machine states
- Data transfer
- Remote control of the TNC

Calling/exiting teleservice

- Select any machine mode of operation.
- Press the MOD key to select the MOD function.

- Establish a connection to the service agency: Set the SERVICE or SUPPORT soft key to ON. The TNC breaks the connection automatically if no new data is transferred for a time set by the machine tool builder (default: 15 min).
- To break the connection to the service agency: Set the SERVICE or SUPPORT soft key to OFF. The TNC terminates the connection after approx. one minute.

13.19 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

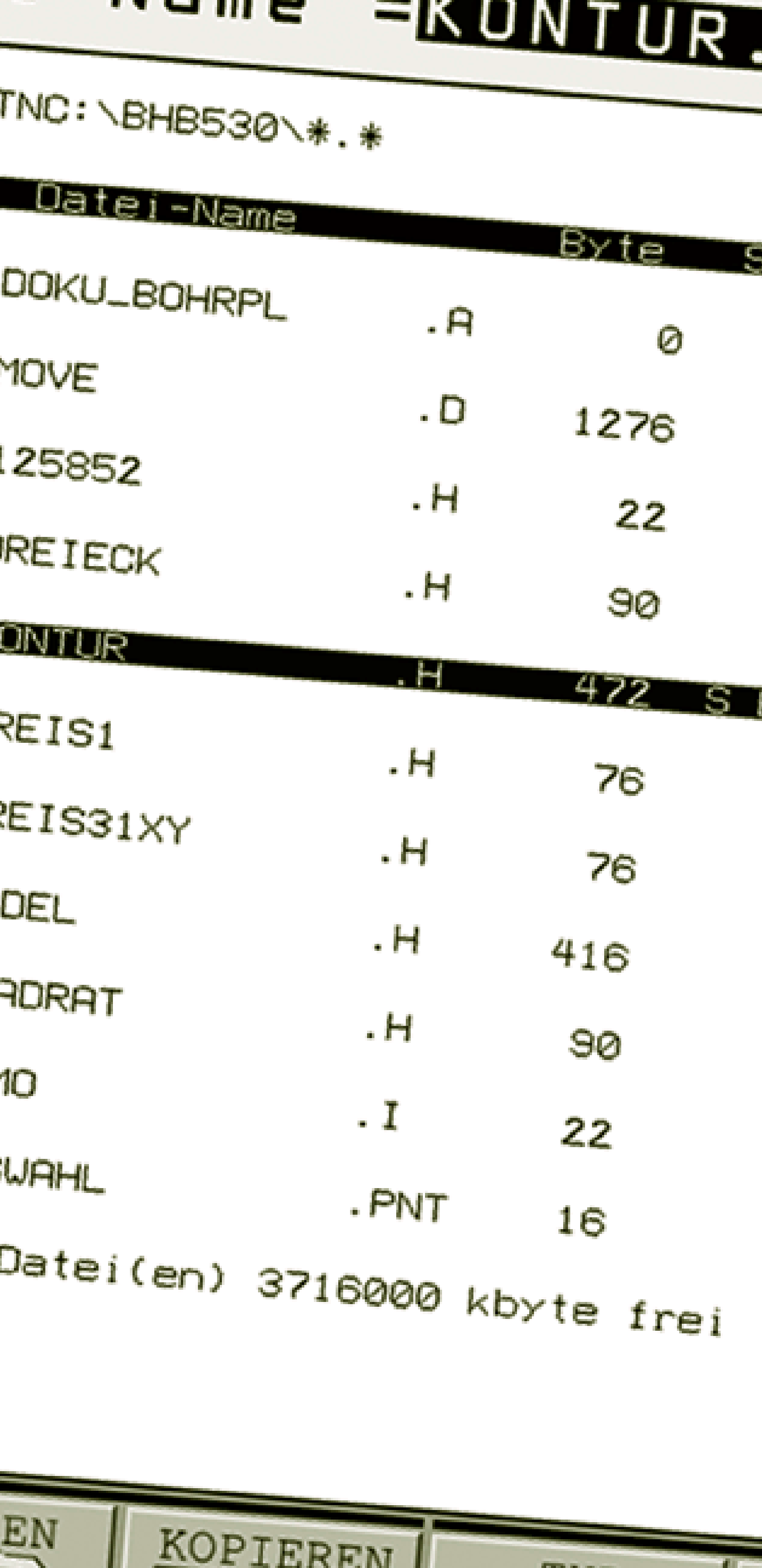
```
REMOTE.TNCPASSWORD=KR1402
REMOTE.TNCPRIVATEPATH=TNC:\RK
```

Permitting/Restricting external access

- Select any machine mode of operation.
- Press the MOD key to select the MOD function.

EXTERNAL ACCESS OFF ON

- 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.



14

Tables and Overviews

14.1 General User Parameters

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

Some 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** Number of stop bits that are transmitted at the end of a character: 1 stop bit: **+0** 2 stop bits: **+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	
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 **3 000** [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	
Pre-position at rapid traverse	**MP6151** Pre-position with speed from **MP6150: 0** Pre-position at rapid traverse: **1**
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** [°]
Automatic operation: Automatically orient the infrared sensor before probing to the programmed probing direction	**MP6165** Function inactive: **0** Orient infrared sensor: **1**
Manual operation: Compensate the probe direction taking an active basic rotation into account	**MP6166** Function inactive: **0** Take basic rotation into account: **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.x (traverse range 1) to MP6181.2 (traverse range3)** **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.x (traverse range 1) to MP6182.2 (traverse range3)** **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.x (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**

3-D Touch Probes	
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**
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 **3 000** [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** **–1**: Function inactive
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	
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**
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**
Locking soft key for tables	**MP7224.2** Do not lock the EDITING ON/OFF soft key: **+0** Lock the EDITING ON/OFF soft key for ■ Without function: **+1** ■ With function: **+2** ■ Tool tables: **+4** ■ Datum tables: **+8** ■ Pallet tables: **+16** ■ With function: **+32** ■ Point tables: **+64**

TNC displays, TNC editor	
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**
Dialog language	**MP7230** 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 (Cyrillic character set): **14** (only on the MC 422 B) Chinese (simplified): **15** (only on the MC 422 B) Chinese (traditional): **16** (only on the MC 422 B) Slovenian: **17** (only on the MC 422 B, **software option**) Norwegian: **18** (only on the MC 422 B, **software option**) Slovak: **19** (only on the MC 422 B, **software option**) Latvian: **20** (only on the MC 422 B, **software option**) Korean: **21** (only on the MC 422 B, **software option**) Estonian: **22** (only on the MC 422 B, **software option**)
Configure tool tables	**MP7260** Inactive: **0** Number of tools generated by the TNC when a new tool table is opened: **1** to **254** If you require more than 254 tools, you can expand the tool table with the function APPEND N LINES see "Tool Data," page 181
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.

TNC displays, TNC editor	
Index tool numbers in order to be able to assign different compensation data to one tool number	**MP7262** Do not index: **0** Number of permissible indices: **1** to **9**
Soft key for pocket tables	**MP7263** Show the POCKET TABLE soft key in the tool table: **0** Do not show the POCKET TABLE soft key in the tool table: **1**
Configure tool table (To omit from the table: enter 0); Column number in the tool table for	**MP7266.0** Tool name – NAME: **0** to **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

TNC displays, TNC editor	
Configure tool table (To omit from the table: enter 0); Column number in the tool table for	**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 **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 **MP7266.32** Limitation of spindle speed – NMAX: **–** to **999999**; Column width: 6 characters **MP7266.33** Retraction at NC stop – LIFTOFF: **Y** / **N**; column width is 1 character **MP7266.34** Machine-dependent function – P1: **–99999.9999** to **+99999.9999**; column width: 10 characters **MP7266.35** Machine-dependent function – P2: **–99999.9999** to **+99999.9999**; column width: 10 characters **MP7266.36** Machine-dependent function – P3: **–99999.9999** to **+99999.9999**; column width: 10 characters **MP7266.37** Tool-specific kinematics description – KINEMATIC: **Name of the kinematics description**; column width: 16 characters **MP7266.38** Point angle – T_ANGLE: **0** to **180**; column width: 9 characters **MP7266.39** Thread pitch PITCH: **0** to **99999.9999**; column width: 10 characters **MP7266.40** Adaptive Feed Control (AFC): **Name of the control setting from the table AFC.TAB**; column width: 10 characters

TNC displays, TNC editor	
Configure tool pocket table (to omit from the table: enter 0); Column number in the pocket table for	**MP7267.0** Tool number – T: **0** to **7** **MP7267.1** Special tool – ST: **0** to **7** **MP7267.2** Fixed pocket – F: **0** to **7** **MP7267.3** Pocket locked – L: **0** to **7** **MP7267.4** PLC status – PLC: **0** to **7** **MP7267.5** Tool name from tool table – TNAME: **0** to **7** **MP7267.6** Comment from tool table – DOC: **0** to **77** **MP7267.7** Tool type – PTYP: **0** to **99** **MP7267.8** Value for PLC – P1: **–99999.9999** to **+99999.9999** **MP7267.9** Value for PLC – P2: **–99999.9999** to **+99999.9999** **MP7267.10** Value for PLC – P3: **–99999.9999** to **+99999.9999** **MP7267.11** Value for PLC – P4: **–99999.9999** to **+99999.9999** **MP7267.12** Value for PLC – P5: **–99999.9999** to **+99999.9999** **MP7267.13** Reserved pocket – RSV: **0** to **1** **MP7267.14** Pocket above locked – LOCKED_ABOVE: **0** to **65535** **MP7267.15** Pocket below locked – LOCKED_BELOW: **0** to **65535** **MP7267.16** Pocket at left locked – LOCKED_LEFT: **0** to **65535** **MP7267.17** Pocket at right locked – LOCKED_RIGHT: **0** to **65535**
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**
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 MP7290.13 (14th 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 in the preset table	**MP7294** 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 in the 10th axis: **+512** Disable datum setting in the 11th axis: **+1024** Disable datum setting in the 12th axis: **+2048** Disable datum setting in the 13th axis: **+4096** Disable datum setting in the 14th axis: **+8192**
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 in the 10th axis: **+512** Disable datum setting in the 11th axis: **+1024** Disable datum setting in the 12th axis: **+2048** Disable datum setting in the 13th axis: **+4096** Disable datum setting in the 14th axis: **+8192**
Disable datum setting with the orange axis keys	**MP7296** Do not disable datum setting: **0** Disable datum setting with the orange axis keys: **1**

TNC displays, TNC editor	
Reset status display, Q parameters, tool data and machining time	**MP7300** Reset all when a program is selected: **0** Reset all when a program is selected and with M02, M30, END PGM (with PGM CALL: END PGM of the highest calling program): **1** Reset only status display and tool data when a program is selected: **2** Reset only status display, machining time and tool data when a program is selected and at M02, M30, END PGM (with PGM CALL: END PGM of the highest calling program): **3** Reset status display, machining time and Q parameters when a program is selected: **4** Reset only status display, machining time and Q parameters when a program is selected and at M02, M30, END PGM (with PGM CALL: END PGM of the highest calling program): **5** Reset status display and machining time when a program is selected: **6** Reset only status display and machining time when a program is selected and at M02, M30, END PGM (with PGM CALL: END PGM of the highest calling program): **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** 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** Software functions of the new 3-D graphics active: **+0** Software functions of the new 3-D graphics inactive: **+16**
Limitation of a tool's tooth length to be simulated. Only effective if LCUTS is not defined.	**MP7312** **0** to **99 999.9999** [mm] Factor by which the tool diameter is multiplied in order to increase the simulation speed. If 0 is entered, the TNC assumes an infinitely long tooth length, which increases the simulation speed.
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 tool axis: M function for end	**MP7317.1** **0** to **88** (0: Function inactive)
Screen saver	**MP7392.0** **0** to **99** [min] Time in minutes until the screen saver switches on (0: Function not active) **MP7392.1** No screen saver is active: **0** Standard screen saver of the X server: **1** 3-D line pattern: **2**

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** The TNC saves the calibrated data for the 3-D touch probe internally: **+0** The TNC uses the compensation values for the touch probe from the tool table as calibration data for the 3-D touch probe: **+1**
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 starting another process: **+0** Mill channel and rough-out for each infeed depth before continuing to the next depth: **+8** The following applies to Cycles 6, 15, 16, 21, 22, 23, and 24: 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, Cycle 5 CIRCULAR POCKET MILLING, and Cycle 6 ROUGH OUT: 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 inactive: **+0** Exact stop for positioning with rotary axes active: **+64**

Machining and program run	
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**
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]
Compatibility machine parameters for datum tables	**MP7475** Datum shifts are referenced to the workpiece datum: **0** If the value **1** was entered in older TNC controls or in controls with software 340 420-xx, datum shifts were referenced to the machine datum. This function is no longer available. You must now use the preset table instead of datum tables referenced to REF (see “Datum management with the preset table” on page 80).

14.2 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices

The interface complies with EN 50 178 for "low voltage electrical separation."

Please note that pins 6 and 8 of the connecting cable 274 545 are bridged.

When using the 25-pin adapter block:

TNC		Connecting cable 365 725-xx			Adapter block 310 085-01		Connecting cable 274 545-xx		
Male	**Assignment**	**Female**	**Color**	**Female**	**Male**	**Female**	**Male**	**Color**	**Female**
1	Do not assign	1		1	1	1	1	WH/BN	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

When using the 9-pin adapter block:

TNC		Connecting cable 355 484-xx			Adapter block 363 987-02		Connecting cable 366 964-xx		
Male	**Assignment**	**Female**	**Color**	**Male**	**Female**	**Male**	**Female**	**Color**	**Female**
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	WH/GN	8	8	8	8	WH/GN	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

Non-HEIDENHAIN devices

The connector pin layout of a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device, depending on the unit and type of data transfer.

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 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	

14.3 Technical Information

Explanation of symbols

■ Standard
□ Axis option
◆ Software option 1
● Software option 2

User functions	
Description	■ Basic version: 3 axes plus spindle ■ Fourth NC axis plus auxiliary axis or □ 8 additional axes or 7 additional axes plus 2nd spindle ■ Digital current and speed control
Programming	HEIDENHAIN conversational format, with smarT.NC and as per ISO
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 up to 30 000 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 and Tool Center Point Management (TCPM) to change the angle of the swivel head during program run without affecting the position of the tool point ● 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
Approaching and departing the contour	■ 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 point 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	■ Mathematical functions =, +, –, *, /, sin α , cos α ■ Logical comparisons (=, =/, <, >) ■ Calculating with parentheses ■ tan α , arcus sin, arcus cos, arcus 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 ■ The context-sensitive help system TNCguide (FCL3 function) ■ 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 B main computer ■ CC 422 or CC 424 controller unit ■ Operating panel ■ 15.1-inch TFT color flat-panel display with soft keys
Program memory	
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 (3 937 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 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
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) ■ USB 2.0 interface For connection of pointing devices (mouse)
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 420** portable handwheel with display or ■ 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 640:** 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 and Tool Center Point Management (TCPM) to change the angle of the swivel head during program run without affecting the position of the tool point ● 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

DXF converter option	
Extracting contour programs from DXF data	■ Format supported: AC1009 (AutoCAD R12) ■ For plain-language and smarT.NC contour programs ■ Simple and convenient specification of reference points

Dynamic collision monitoring (DCM) option	
Collision monitoring in all machine operating modes	■ The machine manufacturer defines objects to be monitored ■ Three warning levels in manual operation ■ Program interrupt during automatic operation ■ Includes monitoring of 5-axis movements

Additional dialog language option	
Additional dialog languages	■ Slovenian ■ Norwegian ■ Slovak ■ Latvian ■ Korean ■ Estonian

Global Program Settings software option	
Function for superimposing coordinate transformations in the Program Run modes	■ Exchanging axes ■ Superimposed datum shift ■ Superimposed mirroring ■ Axis locking ■ Handwheel superimposition ■ Superimposed basic rotation and datum-based rotation ■ Feed rate factor

Adaptive Feed Control software option (AFC)	
Function for adaptive feed-rate control for optimizing the machining conditions during series production	■ Recording the actual spindle power by means of a teach-in cut ■ Defining the limits of automatic feed rate control ■ Fully automatic feed control during program run

Feature content level 2 (FCL 2) option	
Enabling of significant improvements	■ Virtual tool axis ■ Touch probe cycle G441, Rapid Probing ■ Offline CAD point filter ■ 3-D line graphics ■ Contour pocket: Assign a separate depth to each subcontour ■ smarT.NC: Coordinate transformation ■ smarT.NC: PLANE function ■ smarT.NC: Graphically supported block scan ■ Expanded USB functionality ■ Network attachment via DHCP and DNS

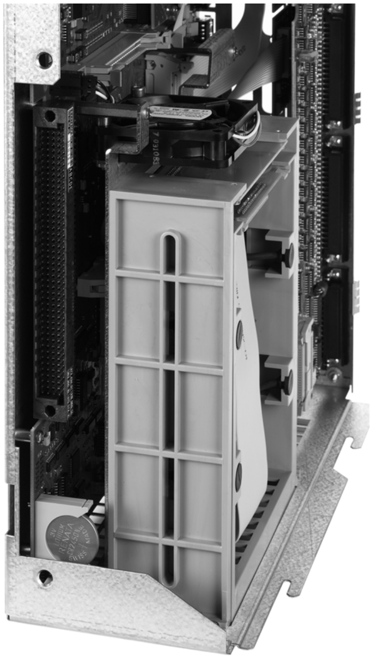
FCL 3 upgrade functions	
Enabling of significant improvements	■ Touch probe cycle for 3-D probing ■ Probing cycles G408 and G409 (Units 408 and 409 in smarT.NC) for setting a reference point in the center of a slot or a ridge ■ PLANE function: Axis angle input ■ User documentation as context-sensitive help right on the TNC. ■ Feed-rate reduction for the machining of contour pockets with the tool being in full contact with the workpiece ■ smarT.NC: Contour pocket on pattern ■ smarT.NC: Parallel programming is possible ■ smarT.NC: Preview of contour programs in the file manager ■ smarT.NC: Positioning strategy for machining point patterns

Input format and unit of TNC functions	
Positions, coordinates, circle radii, chamfer lengths	–99 999.9999 to +99 999.9999 (5.4: places before decimal point, places after decimal point) [mm]
Tool numbers	0 to 32 767.9 (5.1)
Tool names	16 characters, enclosed by quotation marks with TOOL CALL. Permitted special characters: #, $, %, &, -
Delta values for tool compensation	–99.9999 to +99.9999 (2.4) [mm]
Spindle speeds	0 to 99 999.999 (5.3) [rpm]
Feed rates	0 to 99 999.999 (5.3) [mm/min] or [mm/tooth] or [mm/rev]
Dwell time in Cycle 9	0 to 3600.000 (4.3) [s]
Thread pitch in various cycles	–99.9999 to +99.9999 (2.4) [mm]
Angle of spindle orientation	0 to 360.0000 (3.4) [°]
Angle for polar coordinates, rotation, tilting the working plane	-360.0000 to 360.0000 (3.4) [°]
Polar coordinate angle for helical interpolation (CP)	-5 400.0000 to 5 400.0000 (4.4) [°]
Datum numbers in Cycle 7	0 to 2 999 (4.0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2.6)
Miscellaneous functions M	0 to 999 (3.0)
Q parameter numbers	0 to 1999 (4.0)
Q parameter values	-99 999.9999 to +99 999.9999 (5.4)
Labels (LBL) for program jumps	0 to 999 (3.0)
Labels (LBL) for program jumps	Any text string in quotes (“ ”)
Number of program section repeats REP	1 to 65 534 (5.0)
Error number with Q parameter function FN14	0 to 1 099 (4.0)
Spline parameter K	–9.9999999 to +9.9999999 (1.7)
Exponent for spline parameter	-255 to 255 (3.0)
Surface-normal vectors N and T with 3-D compensation	–9.9999999 to +9.9999999 (1.7)

14.4 Exchanging the Buffer Battery

A buffer battery supplies the TNC with current to prevent the data in RAM memory from being lost when the TNC is switched off.

If the TNC displays the error message **`Exchange buffer battery,`** then you must replace the batteries:



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 B

2 Exchange the battery. The socket accepts a new battery only in the correct orientation.

15

iTNC 530 with Windows 2000 (Option)

15.1 Introduction

End User License Agreement (EULA) for Windows 2000

Please pay attention to the Microsoft End User License Agreement (EULA), which is included with your machine documentation.

You can download the EULA from the HEIDENHAIN web site under **www.heidenhain.de**, >**Service**, >**Download Area**, >**Licensing Conditions.**

General Information

The special features of the iTNC 530 with Windows 2000 are described in this chapter. For the Windows 2000 system functions, please refer to the Windows documentation.

The TNC controls from HEIDENHAIN have always been user-friendly: Simple programming in HEIDENHAIN conversational format, field-proven cycles, unambiguous function keys and clearly structured graphic functions make them extremely popular shop-floor programmable controls.

The standard Windows operating system is now also provided as a user interface. The new and highly efficient HEIDENHAIN hardware with two processors is the basis for the iTNC 530 with Windows 2000.

The first processor handles real-time jobs and the HEIDENHAIN operating system, whereas the second processor is available only to the standard Windows operating system and thus provides the user access to the world of information technology.

Again, ease of operation has been given top priority:

- A complete PC keyboard with touchpad is integrated in the operating panel.
- The 15-inch high-resolution color flat-panel monitor displays both the iTNC interface and the Windows applications.
- Standard PC equipment such as a mouse or drives can easily be connected to the control through USB interfaces.

Specifications

Specifications	iTNC 530 with Windows 2000
Version	Dual-processor control with ■ HEROS real-time operating system for controlling the machine ■ Windows 2000 PC operating system as user interface

Memory	■ Random access memory (RAM)
	  ■ 256 MB for control applications
	  ■ 256 MB for Windows applications
	■ Hard disk
	  ■ 13 GB for TNC files
	  ■ 13 GB for Windows files, of which approx. 13 GB are available for applications

Data interfaces	■ Ethernet 10/100 BaseT (up to 100 Mbps depending on network utilization) ■ RS-232-C/V.24 (max. 115 200 bps) ■ RS-422/V.11 (max. 115 200 bps) ■ 2 x USB ■ 2 x PS/2

15.2 Starting an iTNC 530 Application

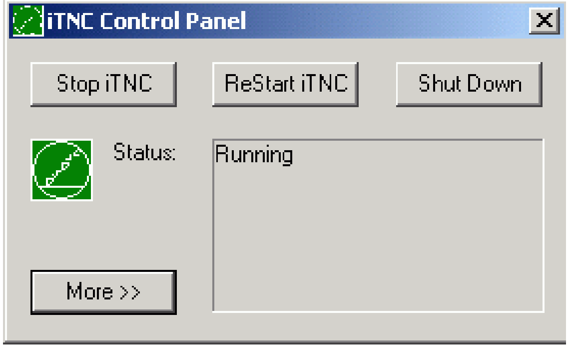
Logging on to Windows

After you have switched on the power supply, the iTNC 530 starts booting automatically. When the input dialog for logging on to Windows appears, there are two possibilities for logging in:

- Logging on as a TNC user
- Logging on as a local administrator

Logging on as a TNC user

- Enter the user name "TNC" in the **`User name`** input box. Leave the **`Password`** input box blank and press the OK button.
- The TNC software is automatically started. The status message **`Starting, please wait...`** appears in the iTNC Control Panel. .



> Do not open or use any other Windows programs as long as the iTNC Control Panel is displayed (see figure). When the iTNC software has successfully started, the Control Panel minimizes itself to a HEIDENHAIN symbol on the task bar.
>
> This user identification permits very limited access to the Windows operating system. You are neither allowed to change the network settings, nor are you allowed to install new software.

Logging on as a local administrator

Please contact your machine tool builder for the user name and the password.

As a local administrator, you are allowed to install software and change the network settings.

HEIDENHAIN does not assist you in installing Windows applications and offers no guarantee for the function of the applications you installed.

HEIDENHAIN is not liable for faulty hard disk contents caused by installing updates to third-party software or additional application software.

If HEIDENHAIN is required to render service after programs or data have been changed, HEIDENHAIN will charge you for the service costs incurred.

In order to guarantee the trouble-free function of the iTNC application, the Windows 2000 system must at all times have sufficient

- CPU performance
- free hard disk memory on the C drive
- RAM
- bandwidth for the hard drive interface

available.

By sufficiently buffering the TNC data, the control can compensate brief interruptions (up to one second at a block cycle time of 0.5 ms) to the data transfer from the Windows PC. However, if the data transfer from the Windows PC is interrupted for a longer time period, problems can occur with the feed rate during program run, resulting in damage to the workpiece.

Keep in mind the following requirements for software installations:

The program to be installed must not overburden the computing power of the Windows PC (256 MB RAM, 266 MHz clock frequency).

Programs executed under Windows with the priority levels **above normal, high** or **real time** (e.g. games), must not be installed.

You should use virus scanners only when the TNC is not running an NC program. HEIDENHAIN recommends using virus scanners either just after switching the control on or just before switching it off.

15.3 Switching Off the iTNC 530

Fundamentals

To prevent data from being lost at switch-off, you must shut down the iTNC 530 properly. The following sections describe the various possibilities for doing so.

Inappropriate switch-off of the iTNC 530 can lead to data loss.

Exit the iTNC 530 application before exiting Windows.

Logging a user off

You can log a user off of Windows at any time without adversely influencing the iTNC software, However, the iTNC screen is not visible during the log-off process, and you cannot make any entries during this time.

Note that machine-specific keys (such as NC Start or the axis direction keys) remain active.

After a new user has logged on, the iTNC screen reappears.

Exiting the iTNC application

Caution!

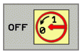
Before you exit the iTNC application, you absolutely must press the Emergency Stop key. Otherwise you could lose data or the machine could become damaged.

There are two possibilities for exiting the iTNC application:

- Internal exiting via the Manual operating mode; simultaneously exits Windows
- External exiting via the iTNC Control Panel; only exits the iTNC application

Internal exiting via the Manual operating mode

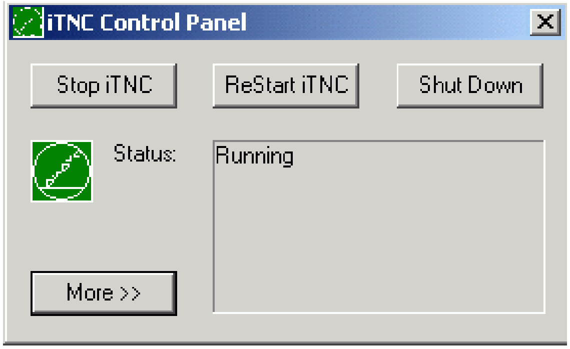
- Select the Manual Operation mode.
- Shift the soft-key row until the soft key for shutting down the iTNC application appears.

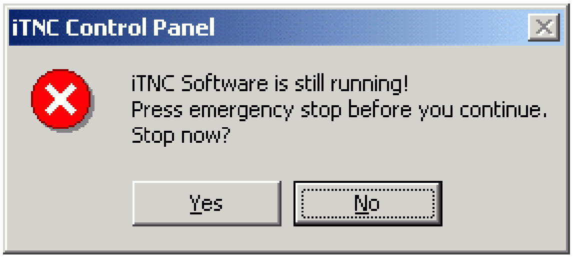


- Select the function for shutting down and confirm the following dialog prompt again with the YES soft key.
- When the message **`It is now safe to turn off your computer.`** appears on the iTNC screen, you may switch off the power supply to the iTNC 530.

External exiting via the iTNC Control Panel

- Press the Windows key on the ASCII keyboard to minimize the iTNC application and display the Task Bar.
- Double-click the green HEIDENHAIN symbol to the lower right in the Task Bar for the iTNC Control Panel to appear (see figure).





- Select the function for exiting the iTNC 530 application: Press the **Stop iTNC** button.
- After you have pressed the Emergency Stop button, acknowledge the iTNC message with the **Yes** screen button. The iTNC application is stopped.
- The iTNC Control Panel remains active. To restart the iTNC 530, press the **Restart iTNC** button.

To exit Windows, select

- the **Start** button
- the menu item **Shut down...**
- again the menu item **Shut down...**
- and confirm with **OK.**

Shutting down Windows

If you try to shut down Windows while the iTNC software is still active, the control displays a warning (see figure).

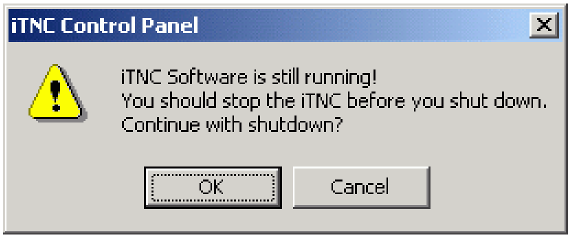
Caution!

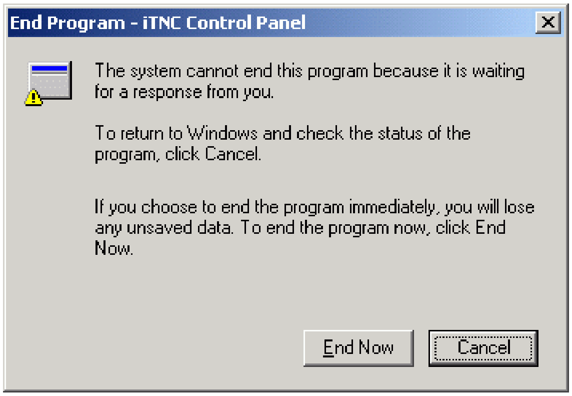
Before you confirm with OK, you absolutely must press the Emergency Stop button. Otherwise you could lose data or the machine could become damaged.

If you confirm with OK, the iTNC software is exited and Windows is shut down.

Caution!

After several seconds Windows displays its own warning, covering the iTNC warning (see figure). Never confirm the warning with End Now, since you could lose data or the machine could become damaged.





15.4 Network Settings

Prerequisite

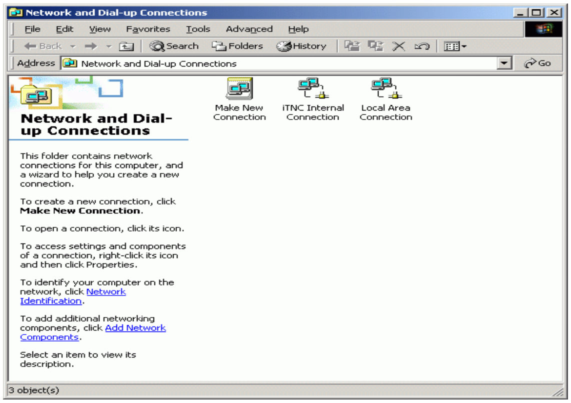
You must log on as a local administrator to be able to change the network settings. Please contact your machine tool builder for the required user name and password.

The network should be configured only by network specialists.

Adjusting the network settings

The iTNC 530 is shipped with two network connections: The **Local Area Connection** and the **iTNC Internal Connection** (see figure).

The **Local Area Connection** is the iTNC's connection to its network. You may adjust all familiar Windows 2000 settings to your network (also see the Windows 2000 network description).



The **iTNC Internal Connection** is exactly that: an internal iTNC connection. The configuration of this connection must not be changed. Changes might cause the iTNC to stop functioning.

This internal network address has a default setting of **`192 168 252 253`** and must not collide with your company network, meaning that the subnet **`192.168.254.xxx`** cannot already exist in your network. If necessary, contact HEIDENHAIN if you are having address conflicts.

The option **`Obtain IP address automatically`** must be inactive.

Controlling access

Administrators have access to the TNC drives D, E and F. Please note that some of the data in these partitions is binary encoded, and write-accesses might lead to undefined behavior of the iTNC.

The partitions D, E and F have access rights for the user groups **SYSTEM** and **Administrators.** The group **SYSTEM** ensures that the Windows service that starts the control has access. The group **Administrators** ensures that the real-time processor of the iTNC receives a network connection via the **iTNC Internal Connection.**

You may not restrict access by these groups, nor may you add groups and prohibit certain accesses by these groups (in Windows, access restrictions have priority over access rights).

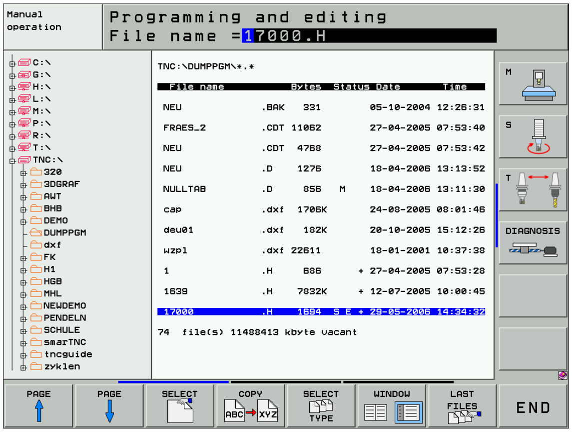
15.5 Specifics About File Management

The iTNC drive

When you call the iTNC file manager, the left window shows all available drives. For example:

- **C:**: Windows partition of the built-in hard disk
- **RS232:**: Serial interface 1
- **RS422:**: Serial interface 2
- **TNC:**: Data partition of the iTNC

There might also be other networks available if you have connected them with Windows Explorer.



> Please note that the data drive of the iTNC appears in the file manager with the name **TNC:**. In Windows Explorer, this drive (partition) appears with the letter **D.**
>
> Subdirectories on the TNC drive (e.g. **RECYCLER** and **SYSTEM VOLUME IDENTIFIER**) are created by Windows 2000 and may not be deleted.
>
> With machine parameter 7225 you can define letters of drives that should not be shown in the file management of the TNC.

If you connect a new network drive with Windows Explorer, you may have to update the iTNC's display of available drives:

- ▶ To call the file manager, press the PGM MGT key.
- ▶ Move the highlight to the drive window at left.
- ▶ Switch to the second level of the soft-key row.
- ▶ To update the drive overview, press the UPDATE TREE soft key.

Data transfer to the iTNC 530

Before you can initiate a data transfer on the iTNC, the network drive must have been connected with Windows Explorer. Access to UNC network names (e.g. \\PC0815\DIR1) is not possible.

TNC-specific files

After integrating the iTNC 530 into your network, you can access any computer and transfer files to it from the iTNC. However, certain file types may only be transferred if the data transfer was initiated by the iTNC. The reason is that these files must be converted into binary format during the data transfer to the iTNC.

Simply copying the file types listed below to the D drive using Windows Explorer is both prohibited and useless.

File types that may not be copied using Windows Explorer:

- Conversational dialog programs (extension .H)
- smarT.NC unit programs (extension .HU)
- smarT.NC contour programs (extension .HC)
- ISO programs (extension .I)
- Tool tables (extension .T)
- Pocket tables (extension .TCH)
- Pallet tables (extension .P)
- Datum tables (extension .D)
- Point tables (extension .PNT)
- Cutting data tables (extension .CDT)
- Freely definable tables (extension .TAB)

Procedure for data transfer: See "Data transfer to or from an external data medium," page 123.

ASCII files

There are no limitations regarding the direct copying of ASCII files (files with the extension .A) with Windows Explorer.

Please note that all the files you want to use on the TNC must be stored on drive D.

SYMBOLE

A

B

C

C

D

E

F

Table of Miscellaneous Functions

M	Effect	Effective at block...	Start	End	Page
M00	Stop program/Spindle STOP/Coolant OFF			■	Page 251
M01	Optional program STOP			■	Page 575
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1			■	Page 251
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP		■ ■ 	 ■	Page 251
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP			■	Page 251
M08 M09	Coolant ON Coolant OFF		■ 	 ■	Page 251
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON		■ ■		Page 251
M30	Same function as M02			■	Page 251
M89	Vacant miscellaneous function **or** Cycle call, modally effective (depending on machine parameter)		■ 	 ■	Page 285
M90	Only in lag mode: Constant contouring speed at corners			■	Page 255
M91	Within the positioning block: Coordinates are referenced to machine datum		■		Page 252
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position		■		Page 252
M94	Reduce display of rotary axis to value under 360°		■		Page 272
M97	Machine small contour steps			■	Page 257
M98	Machine open contours completely			■	Page 259
M99	Blockwise cycle call			■	Page 285
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Cancel M101		■ 	 ■	Page 195
M103	Reduce feed rate during plunging to factor F (percentage)		■		Page 260
M104	Reactivate the datum as last defined		■		Page 254
M105 M106	Machining with second k_V factor Machining with first k_V factor		■ ■		Page 640
M107 M108	Suppress error message for replacement tools Cancel M107		■ 	 ■	Page 194

M	Effect	Effective at block... Start	End	Page
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)	■		Page 262
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)	■		
M111	Cancel M109/M110		■	
M114	Automatic compensation of machine geometry when working with tilted axes	■		Page 273
M115	Reset M114		■	
M116	Feed rate for angular axes in mm/minn	■		Page 270
M117	Cancel M116		■	
M118	Superimpose handwheel positioning during program run	■		Page 264
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	■		Page 262
M124	Do not include points when executing non-compensated line blocks	■		Page 256
M126	Shortest-path traverse of rotary axes	■		Page 271
M127	Cancel M126		■	
M128	Maintain the position of the tool tip when positioning with tilted axes (TCPM)	■		Page 274
M129	Cancel M128		■	
M130	Moving to position in an untilted coordinate system with a tilted working plane	■		Page 254
M134	Exact stop at nontangential contour transitions when positioning with rotary axes	■		Page 276
M135	Reset M134		■	
M136	Feed rate F in millimeters per spindle revolution	■		Page 261
M137	Reset M136		■	
M138	Select tilting axes	■		Page 276
M140	Retraction from the contour in the tool-axis direction	■		Page 265
M141	Suppress touch probe monitoring	■		Page 266
M142	Delete modal program information	■		Page 267
M143	Delete basic rotation	■		Page 267
M144	Compensating the machine’s kinematic configuration for ACTUAL/NOMINAL positions at end of block	■		Page 277
M145	Cancel M144		■	
M148	Automatically retract tool from the contour at an NC stop	■		Page 268
M149	Cancel M148		■	
M150	Suppress limit switch message (function effective blockwise)	■		Page 269
M200	Laser cutting: Output programmed voltage directly	■		Page 278
M201	Laser cutting: Output voltage as a function of distance	■		
M202	Laser cutting: Output voltage as a function of speed	■		
M203	Laser cutting: Output voltage as a function of time (ramp)	■		
M204	Laser cutting: Output voltage as a function of time (pulse)	■		

ISO Function Overview of the iTNC 530

M Functions	
M00 M01 M02	Stop program run/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 Cancel 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 Cancel 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) Cancel M109/M110

M Functions	
M114 M115	Automatic compensation of machine geometry when working with tilted axes Cancel M114
M116 M117	Feed rate for angular axes in mm/min Cancel 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 Cancel M126
M128 M129	Maintain the position of the tool tip when positioning with tilted axes (TCPM) Cancel 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 Cancel M134
M136 M137	Feed rate F in millimeters per spindle revolution Cancel 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 Cancel M144
M150	Suppress limit switch message
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, clockwise
G03	Circular interpolation, Cartesian coordinates, counterclockwise
G05	Circular interpolation, Cartesian coordinates, without indication of direction
G06	Circular interpolation, Cartesian coordinates, tangential contour approach
G07*	Paraxial positioning block
G10	Straight-line interpolation, polar coordinates, rapid traverse
G11	Straight-line interpolation, polar coordinates
G12	Circular interpolation, polar coordinates, clockwise
G13	Circular interpolation, polar coordinates, counterclockwise
G15	Circular interpolation, polar coordinates, without indication of direction
G16	Circular interpolation, polar coordinates, tangential contour approach

Chamfer/Rounding/Approach contour/Depart contour

G24*	Chamfer with length R
G25*	Corner rounding with radius R
G26*	Tangential contour approach with radius R
G27*	Tangential contour approach with radius R

Define the tool

G99*	With tool number T, length L, radius R

Tool radius compensation

G40	No tool radius compensation
G41	Tool radius compensation, left of the contour
G42	Tool radius compensation, right of the contour
G43	Paraxial compensation for G07, lengthening
G44	Paraxial compensation for G07, shortening

Blank form definition for graphics

G30	(G17/G18/G19) min. point
G31	(G90/G91) max. point

Cycles for Drilling, Tapping and Thread Milling

G240	Centering
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

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
G251	Rectangular pocket
G252	Circular pocket
G253	Slot
G254	Circular slot

Cycles for creating point patterns

G220	Circular point pattern
G221	Point patterns on lines

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	3-D data
G230	Multipass milling of plane surfaces
G231	Multipass milling of tilted surfaces

*) Non-modal function

Touch probe cycles for measuring workpiece misalignment

G400	Basic rotation using 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

G functions

Touch probe cycles for datum setting

G408	Slot center reference point
G409	Reference point at center of hole
G410	Reference point from inside of rectangle
G411	Datum from outside of rectangle
G412	Reference point from inside of circle
G413	Reference point from outside of circle
G414	Reference point in outside corner
G415	Reference point in inside corner
G416	Reference point circle center
G417	Reference point in touch probe axis
G418	Reference point in center of 4 holes
G419	Reference point in selectable axis

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	Calibrate 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
G441	Fast probing

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
Machining plane	PLANE ...	RESET PLANE

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 02
533 188-22 · SW03 · 3 · 10/2006 · F&W · Printed in Germany · Subject to change without notice