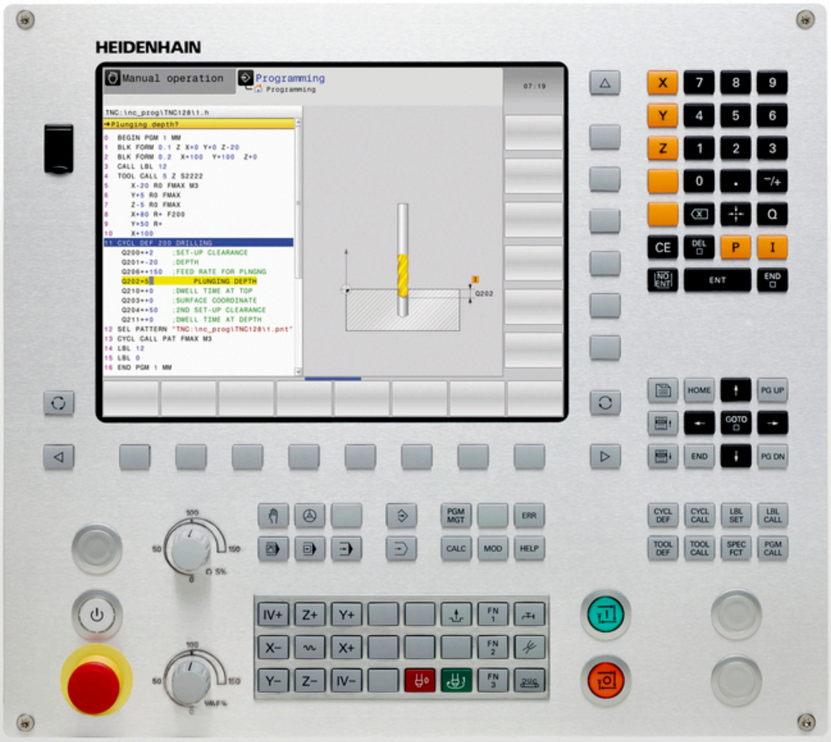
HEIDENHAIN



TNC 128

User's Manual
Conversational Programming

NC Software
771841-06

English (en)
10/2017

Controls and displays

Keys

Keys on visual display unit

Key	Function
	Selecting the screen layout
	Toggle the display between machine operating mode, programming mode, and a third desktop
	Soft keys for selecting functions on screen
◁ ▷ △	Shifting between soft-key rows

Machine operating modes

Key	Function
	Manual operation
	Electronic handwheel
	Positioning with manual data input
	Program run, single block
	Program run, full sequence

Programming modes

Key	Function
	Programming
	Test run

Entering and editing coordinate axes and numbers

Key	Function
X . . . V	Select coordinate axes or enter them in a program
0 . . . 9	Numbers
. −/+	Decimal separator / Reverse algebraic sign
P I	Polar coordinate entry / Incremental values
Q	Q parameter programming / Q parameter status
	Capture actual position
NO ENT	Skip dialog questions, delete words
ENT	Confirm entry and resume dialog
END	Conclude block and exit entry
CE	Clear entries or error message
DEL	Abort dialog, delete program section

Tool functions

Key	Function
TOOL DEF	Define tool data in the program
TOOL CALL	Call tool data

Managing programs and files, control functions

Key	Function
PGM MGT	Select or delete programs and files, external data transfer
PGM CALL	Define program call, select datum and point tables
MOD	Select MOD functions
HELP	Display help text for NC error messages, call TNCguide
ERR	Display all current error messages
CALC	Show calculator
SPEC FCT	Show special functions

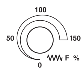
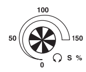
Navigation keys

Key		Function
↑	←	Position the cursor
GOTO □		Go directly to blocks, cycles and parameter functions
HOME		Navigate to the program start or table start
END		Navigate to the program end or end of a table line
PG UP		Navigate up one page
PG DN		Navigate down one page
		Select the next tab in forms
		Up/down one dialog box or button

Cycles, subprograms and program section repeats

Key		Function
CYCL DEF	CYCL CALL	Define and call cycles
LBL SET	LBL CALL	Enter and call labels for subprogramming and program section repeats

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed
	

Fundamentals

About this manual

Safety precautions

Comply with all safety precautions indicated in this document and in your machine tool builder's documentation!

Precautionary statements warn of hazards in handling software and devices and provide information on their prevention. They are classified by hazard severity and divided into the following groups:

⚠ DANGER

Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard will **result in death or severe injury.**

⚠ WARNING

Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury.**

⚠ CAUTION

Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in things other than personal injury, like property damage**.

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

- Signal word indicating the hazard severity
- Type and source of hazard
- Consequences of ignoring the hazard, e.g.: "There is danger of collision during subsequent machining operations"
- Escape – Hazard prevention measures

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.
In these instructions, you will find the following informational notes:

	The information symbol indicates a **tip**. A tip provides important additional or supplementary information.

	This symbol prompts you to follow the safety precautions of your machine tool builder. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.

	The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine tool builder or other supplier.

Would you like any changes, or have you found any errors?

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address:

tnc-userdoc@heidenhain.de

Control model, software and features

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

Control model	NC software number
TNC 128	771841-06
TNC 128 Programming Station	771845-06

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

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

- Probing functions for the 3-D touch probe

In order to find out about the actual features of your machine, please contact the machine manufacturer.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the HEIDENHAIN controls. Participation in one of these courses is recommended to familiarize yourself thoroughly with the control's functions.

Software options

The TNC 128 features various software options that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Additional Axis (option 0 and option 1)	
Additional axis	Additional control loops 1 and 2
Touch Probe Functions (option 17)	
Touch probe functions	**Touch probe cycles:** ■ Presetting in the **Manual operation** mode of operation ■ Tools can be measured automatically
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component

Feature Content Level (upgrade functions)

Along with software options, significant further improvements of the control software are managed via the **F**eature **C**ontent **L**evel upgrade functions. If you install a software update on your control you do not automatically have the functions available as covered by the FCL.

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

Upgrade functions are identified in the manual as **FCL n**. The **n** signifies the serial number of the development status.

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

Intended place of operation

The control 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** operating mode
- MOD function
- **LICENSE INFO** soft key

New functions

New functions 77184x-02

- New special operating mode RETRACT, see "Retraction after a power interruption", page 454
- New graphic simulation, see "Graphics ", page 432
- New **Tool usage file** MOD function in the machine settings group, see "Tool usage file", page 472
- New **Set system time** MOD function in the systems settings group, see "Set the system time", page 474
- New **Graphic settings** MOD group, see "Graphic settings", page 468
- With the new cutting data calculator you can calculate the spindle speed and the feed rate, see "Cutting data calculator", page 164
- With the jump commands new if/then decisions have been introduced, see "Programming if-then decisions", page 253
- New Cycle 233 Face Milling, see "FACE MILLING (Cycle 233)", page 583
- In the drilling cycles 200, 203 and 205 the parameter Q395 DEPTH REFERENCE has been introduced in order to evaluate the T ANGLE

Modified functions 77184x-02

- Up to 4 M functions are now allowed in an NC block, see "Fundamentals", page 336
- New soft keys for transferring values have been introduced in the pocket calculator, see "Operation", page 161
- The distance-to-go display can now also be displayed in the input system, see "Select the position display", page 475
- Several input parameters have been added to Cycle 241 SINGLE-LIP DEEP HOLE DRILLING, see "SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)", page 551
- In the thread milling cycles 26x an approaching feed rate has been introduced
- In Cycle 205 Universal Pecking you can now use parameter Q208 to define a feed rate for retraction, see "Cycle parameters", page 546

New functions 77184x-03

- Programs with .HU and .HC extensions can be selected and processed in all operating modes
- The functions **SELECT PROGRAM** and **CALL SELECTED PROGRAM** have been introduced, see "Calling any program as a subprogram", page 229
- New **FEED DWELL** function for programming repeating dwell times, see "Dwell time FUNCTION FEED", page 358
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", page 268
- USB data carriers can be locked with the SELinux security software, see "SELinux security software", page 99
- The machine parameter **posAfterContPocket** (no. 201007) that influences positioning after an SL cycle has been introduced, see "Machine-specific user parameters", page 640
- Protective zones can be defined in the MOD menu, see "Entering traverse limits", page 471
- Write protection is possible for individual lines in the preset management, see "Saving presets in the table", page 396
- New manual probing function for aligning a plane, see "Measuring 3-D basic rotation"
- CAD files can be opened without option number 42, see "Data Transfer from CAD Files", page 217

Modified functions 77184x-03

- FZ and FU feed rate input possible in the Tool Call block, see "Calling the tool data", page 202
- The input range of the DOC column in the pocket table has been expanded to 32 characters, see "Pocket table for tool changer", page 199
- Commands FN 15, FN 31, FN 32, FT and FMAXT from predecessor controls no longer generate ERROR blocks during import. When simulating or running an NC program with these commands, the control interrupts the NC program with an error message that helps you to find an alternative implementation
- The maximum file size of files output with FN 16: F-PRINT has been increased from 4 KB to 20 KB
- The Preset.PR preset management is write-protected in Programming operating mode, see "Saving presets in the table", page 396
- The input range of the Q parameter list for defining the QPARA tab on the status display consists of 132 input positions, see "Displaying Q parameters (QPARA tab)", page 90
- Manual calibration of the touch probe with fewer pre-positioning movements, see "Calibrating 3-D touch probes (option 17)", page 412
- The position display takes into account the DL oversizes programmed in the Tool Call block, selectable as an oversize of the workpiece or tool, see "Delta values for lengths and radii", page 187
- In single block mode the control executes each point individually with point pattern cycles and CYCL CALL PAT, see "Program run", page 447
- Rebooting the control is no longer possible with the **END** key but with the **RESTART** soft key, see "Switch-off", page 379
- The control displays the contouring feed rate in manual mode, see "Spindle speed S, feed rate F and miscellaneous function M", page 394
- Deactivate tilting in manual mode is only possible via the 3D-ROT menu, see "Activating manual tilting:"
- Machine parameter **maxLineGeoSearch** (no. 105408) has been increased to max. 50000, see "Machine-specific user parameters", page 640

New and modified cycle functions 77184x-03

- Cycle **253 SLOT MILLING** has been added (option 19), see "SLOT MILLING (Cycle 253, DIN/ISO: G253)", page 575

New functions 77184x-05

- New function **FUNCTION DWELL** for programming a dwell time, see "Dwell time FUNCTION DWELL", page 373
- New function **FUNCTION S-PULSE** for programming pulsing shaft speeds, see "Pulsing spindle speed FUNCTION S-PULSE", page 356
- The column **KINEMATIC** has been added to the tool table, see "Entering tool data into the table", page 188
- The column **OVRTIME** has been added to the tool table, see "Entering tool data into the table", page 188
- When importing tool data the CSV file may contain additional table columns not recognized by the control. During import a message is displayed indicating the unrecognized columns and informing that these values will not be adopted, see "Importing and exporting tool data"
- Manual probe functions create a line in the preset management if the specified line does not exist yet, see "Writing measured values from the touch-probe cycles to the preset table", page 411
- Manual probe functions can write in a password-protected line, see "Recording measured values from the touch probe cycles", page 410
- During a manual touch probe cycle, control can be transferred to the handwheel, see "Traverse movements with a handwheel with display", page 406
- Several handwheels can be connected to a control, see "Traverse with electronic handwheels"
- In **Electronic handwheel** mode of operation, the handwheel axis for an HR 130 can be selected with the orange axis keys
- If the control is set to the INCH unit of measure, the control also includes movements traversed by the handwheel in INCHES, see "Traverse with electronic handwheels"
- The control differentiates between interrupted or stopped NC programs. In the interrupted status, the control offers more intervention options, see "Interrupting, stopping or aborting machining", page 449
- With active structuring the structure block can be edited in the structure window, see "Definition and applications", page 159
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", page 268
- The FN16 functions have been expanded, see "FN16: F-PRINT – Formatted output of texts and Q parameter values", page 261
- The file saved with **SAVE AS** is now also found in the file management under **LAST FILES**, see "Editing an NC program", page 119
- If you save files with **SAVE AS**, you can select the target directory with the **SWITCH** soft key, see "Editing an NC program", page 119
- It is possible to search quickly for a file in file management by entering the first letter, see "Selecting drives, directories and files", page 131

- File management displays vertical scrollbars and supports scrolling with the mouse, see "Calling the file manager", page 130
- New machine parameter for recreating **M7** and **M8**, see "Machine-specific user parameters", page 640
- The function **STRLEN** checks whether a string parameter has been defined, see "Finding the length of a string parameter", page 326
- The function **SYSSTR** enables the NC software version to be read out, see "Reading system data", page 323
- The function **FN 38: SEND** can now be programmed without a code number
- Undefined Q parameters can now be transferred with the function **FN 0**
- For jumps with **FN 9**, QS parameters and texts are permitted as conditions, see "Programming if-then decisions", page 253
- Cylindrical workpiece blanks can now also be defined with a diameter instead of a radius, see "Defining the blank: BLK FORM", page 113
- In operating modes **Program run, single block** and **Program run, full sequence** the screen layout **PROGRAM + SECTS** can be specified, see "Structuring programs", page 159
- In operating modes **Program Run Full Sequence**, **Program Run Single Block** and **Positioning w/ Manual Data Input**, the font size can be set to the same size as the **Programming** operating mode, see "Machine-specific user parameters", page 640
- The functions in the **Positioning w/ Manual Data Input** mode were expanded and adapted for improved operation, see "Positioning with Manual Data Input", page 425
- In the **RETRACT** operating mode, feed-rate limitation can be deactivated with the **CANCEL THE FEED RATE LIMITATION** soft key, see "Retraction after a power interruption", page 454
- In the **Test Run** operating mode a tool usage file can also be created without simulation, see "Tool usage test", page 205
- In the **Test Run** operating mode you can hide the rapid traverse movements with the **FMAX PATHS** soft key, see "3-D view in the Test Run operating mode", page 437

- In the **Test Run** operating mode you can reset the solid-model view with the **RESET THE VOLUME MODEL** soft key, see "3-D view in the Test Run operating mode", page 437
- In the **Test Run** operating mode you can reset the tool paths with the **RESET TOOL PATHS** soft key, see "3-D view in the Test Run operating mode", page 437
- In the **Test Run** operating mode the **MEASURING** soft key displays the coordinates if you position the mouse on the graphics, see "3-D view in the Test Run operating mode", page 437
- In the **Test Run** operating mode the **STOP AT** soft key simulates up to a predefined block, see "Test Run up to a certain block ", page 446
- Active basic transformation is shown in the status display on the **POS** tab, see "Positions and coordinates (POS tab)", page 89
- The status display now also shows the path of the active main program, see "Overview", page 87, see "General program information (PGM tab)", page 88
- Mid-program startup can now be continued, see "Entering the program at any point: Mid-program startup", page 457
- With functions **NC/PLC Backup** and **NC/PLC Restore** you can save and restore single directories or the complete TNC drive, see "Backup and restore", page 102
- The new HR 520 and HR 550FS handwheels are supported, see "Moving with the electronic display handwheels", page 383

Modified functions 77184x-05

- When editing the tool table or tool management, only the current table line is blocked, see "Editing the tool table", page 193
- When importing tool tables, nonexistent tool types are imported as type undefined, see "Importing tool tables", page 197
- Tool names can now also include the special characters **%** and **,**, see "Tool number, tool name", page 186
- When importing tool tables the numerical values are adopted from the **R-OFFS** column, see "Importing tool tables", page 197
- In the **LIFTOFF** column of the tool table the default is now **N**, see "Entering tool data into the table", page 188
- The **L** and **R** columns of the tool table are empty when a new tool is created, see "Editing the tool table", page 193
- In the tool table, the **SELECT** soft key is now available for the **RT** and **KINEMATIC** columns, see "Entering tool data into the table", page 188
- You cannot delete the tool data of tools still stored in the pocket table, see "Editing the tool table", page 193
- In all manual probing functions, quicker selection of the start angle of holes and studs is possible with soft keys (paraxial probing directions), see "Functions in touch probe cycles", page 407
- When probing, after acceptance of the actual value of the 1st point, for the 2nd point the soft key for the axis direction is shown
- In all manual probing functions, the direction of the reference axis is suggested as a default
- In manual probing cycles the hard keys **END** and **Adopt Actual Position** may be used
- The display of the machining feed rate has been changed in manual mode, see "Spindle speed S, feed rate F and miscellaneous function M", page 394
- The **FMAX** soft key in Program Run not only limits the machining feed rate during execution of the program but also the axis feed rate for manual axis movements, see "Feed rate limit F MAX", page 395
- Soft key allocations were adapted for incremental positioning
- The values entered for the traverse limits are checked for validity, see "Entering traverse limits", page 471

- When the preset management is opened, the cursor is on the line of the active preset
- The feed rate potentiometer only reduces the programmed feed rate and no longer the feed rate calculated by the control, see "Feed rate F", page 184
- Block editing no longer causes block marking to be canceled. If a block is edited with active block marking and another block is then selected via the syntax search, the marking is expanded to the newly selected block, see "Marking, copying, cutting and inserting program sections", page 122
- The current structure block can be more clearly recognized in the structure window, see "Definition and applications", page 159
- DHCP Lease Time is now also valid following power interruption. When HEROS is shut down, the DHCP server is no longer informed that the IP address is free again, see "Configuring the control", page 486
- In the status display the fields for the LBL names have been expanded to 32 characters
- The **TT** status display now also shows values if the user changes to the **TT** tab later
- Status displays can now also be switched over with the **Next tab** key, see "Additional status displays", page 87
- If a subprogram called with **CALL PGM** ends with **M2** or **M30** the control outputs a warning
- **M124** no longer triggers an error message but only a warning. This enables NC programs with programmed **M124** to run through without interruption
- In the file management, the programs or directories at the cursor position are also displayed in a separate field beneath the current path display
- Upper and lower cases for a file name can be modified in the file management
- If a larger file is transferred to a USB device in the file management, the control displays a warning until file transfer is completed, see "USB devices on the control", page 151
- In the file management, the control also shows the momentary type filter with the path
- In the file management the **SHOW ALL** soft key is now displayed in all operating modes
- In the file management the function **Select the target directory** was modified for copying files or directories. The soft keys **OK** and **CANCEL** are available on the first two positions

- The colors of the programming graphics were changed, see "Programming graphics", page 166
- In the **Test Run** and **Programming** operating modes the tool data is reset when a program is reselected or restarted with the **RESET + START** soft key
- In the **Test Run** operating mode the control displays the datum of the machine table as the reference point when using **BLANK IN WORK SPACE**, see "Showing the workpiece blank in the working space", page 442
- After modification of the active preset, resuming the program is only possible after **GOTO** or mid-program startup, see "Moving the machine axes during an interruption", page 452
- Mid-program startup operation and dialog guidance has been improved, also for pallet tables, see "Entering the program at any point: Mid-program startup", page 457

New and modified cycle functions 77184x-05

- In Cycle 247 PRESETTING, the preset number from the preset table can be selected with the corresponding parameter, see "PRESETTING (Cycle 247)", page 601
- With Cycles 200 and 203 the behavior of the dwell time at top has been adapted, see "UNIVERSAL DRILLING (Cycle 203)", page 534
- Cycle 205 performs deburring on the coordinate surface, see "UNIVERSAL PECKING (Cycle 205)", page 544
- In Cycles 481 to 483, parameter Q340 was expanded with the input option "2". This makes it possible to check the tool without changing the tool table, see "Cycle parameters", page 633, see "Cycle parameters", page 635, see "Cycle parameters", page 637

New functions 77184x-06

- New **FUNCTION COUNT** function for controlling a counter, see "Defining a counter", page 349
- You can also open the tool-carrier files in the file management, see "Tool carrier management", page 368
- With the **ADAPT NC PGM / TABLE** function, you can also import and modify freely definable tables, see "Importing tool tables", page 197
- The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs when importing a table, see "Importing tool tables", page 197
- A quick search for the tool name is possible in the tool table, see "Entering tool data into the table", page 188
- It is possible to comment out NC blocks, see "Commenting out an existing NC block", page 156
- The machine tool builder can disable the setting of presets in individual axes, see "Saving presets in the table", page 396, see "Presetting with a 3-D touch probe (option number 17)", page 418
- Line 0 of the preset table can also be edited manually, see "Saving presets in the table", page 396
- When multiple instances of the CAD viewer are open, they are shown somewhat smaller on the third desktop.
- The nodes in all tree structures can be expanded and collapsed by double-clicking them.
- New icon in the status display for mirrored machining, see "General status display", page 85
- Graphic settings in the **Test Run** operating mode are permanently stored, see "3-D view in the Test Run operating mode", page 437
- In the **Test Run** operating mode, you can now choose between various traverse ranges, see "Application", page 442
- With the **TCH PROBE MONITOR OFF** soft key you can suppress touch-probe monitoring for 30 seconds, see "Suppress touch probe monitoring", page 407
- If the function for orienting the touch probe to the programmed probe direction is active, the number of spindle revolutions is limited when the guard door is open. In some cases, the direction of spindle rotation will change so that positioning will not always follow the shortest path.
- With FN 16: F-PRINT, it is possible to enter references to Q parameters or QS parameters as the source and target, see "FN16: F-PRINT – Formatted output of texts and Q parameter values", page 261
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", page 268

- New machine parameter **iconPrioList** (no. 100813) for defining the order of icons in the status display, see "Machine-specific user parameters", page 640
- The machine parameter **clearPathAtBlk** (no. 124203) enables you to specify whether the tool paths will be cleared with a new BLK FORM in the **Test Run** operating mode, see "Machine-specific user parameters", page 640

Modified functions 77184x-06

- If you use locked tools, the control displays a warning in the **Programming** and **Test Run** operating modes, see "Programming graphics", page 166, see "Test run", page 444
- The control offers a positioning logic for returning to the contour, see "Returning to the contour", page 462
- The positioning logic for returning to the contour with a replacement tool has changed, see "Tool change", page 204
- If the control finds a stored interruption point on restart, you can resume the machining operation from that point, see "Entering the program at any point: Mid-program startup", page 457
- The **TRANS DATUM AXIS** NC syntax can also be used within a contour in the SL cycle.
- Holes and threads are shown in light blue in the programming graphics, see "Programming graphics", page 166
- The tool is shown in red in the graphics while it is in contact with the workpiece, and blue during air cuts, see "Tool display", page 440
- The positions of the sectional planes are no longer reset when a program or a new blank form is selected, see "Projection in three planes", page 435
- Spindle speeds can be entered with decimal places also in the **Manual operation** mode. The control displays the decimal places when the spindle speed is < 1000, see "Entering values", page 394
- The sort order and the column widths in the tool selection window are retained when the control is switched off, see "Calling the tool data", page 202
- If a file to be deleted does not exist, **FILE DELETE** no longer generates an error message.
- If a subprogram called with CALL PGM ends with **M2** or **M30**, the control issues a warning. The control automatically clears the warning as soon as you select another NC program, see "Programming notes", page 228
- The control displays an error message in the header until it is cleared or replaced by a higher-priority error, see "Display of errors", page 170
- The time needed to paste a large amount of data into an NC program was considerably reduced.
- To connect a USB stick you no longer have to press a soft key, see "Connecting and removing USB storage devices", page 140
- The speed of setting the jog increment, spindle speed and feed rate was adjusted for electronic handwheels.
- The control automatically recognizes whether a table is to be imported or the table format is to be adapted, see "Importing tool tables", page 197
- When you double-click a selection field of the table editor with the mouse or press the **ENT** key, a pop-up window opens.
- When configuration subfiles are modified, the control no longer aborts the test run, but only displays a warning.

- You can neither set nor modify a preset without having referenced the axes, see "Traverse reference points", page 378
- The control issues a warning if the handwheel potentiometers are still active when the handwheel is deactivated, see "Moving with the electronic display handwheels", page 383
- When using the HR 550 or HR 550FS handwheels, a warning is issued if the battery voltage is too low, see "Moving with the electronic display handwheels", page 383
- The machine tool builder can define whether the **R-OFFS** offset will be taken into account for a tool with **CUT** 0, see "Tool table: Tool data required for automatic tool measurement", page 192
- The machine tool builder can change the simulated tool change position, see "Test run", page 444
- In the machine parameter **decimalCharakter** (no. 100805) you can define whether a period or a comma will be used as the decimal separator, see "Machine-specific user parameters", page 640

New and modified cycle functions 77184x-06

- Cycles 256 **RECTANGULAR STUD** were extended by the parameters Q215, Q385, Q369 and Q386. see "RECTANGULAR STUD (Cycle 256)", page 579
- Changes of details in Cycle 233: Monitors the tooth length (**LCUTS**) during finishing, increases the area by Q357 in the milling direction when roughing with milling strategies 0 to 3 (provided that no limit has been set in the milling direction) see "FACE MILLING (Cycle 233)", page 583
- The technologically outdated Cycles 1, 2, 3, 4, 5, 17, 212, 213, 214, 215, 210, 211, 230, and 231 grouped under **OLD CYCLES** can no longer be inserted using the editor. These cycles can still be executed and edited, however.
- The tool touch probe cycles, such as Cycles 480, 481 and 482, can be hidden see "Machine-specific user parameters", page 640
- New SERIAL column in the touch probe table see "touch probe data", page 623

Contents

1

First Steps with the TNC 128

1.1 Overview

This chapter is intended to help users quickly learn to handle the most important procedures on the control. For more information on a respective topic, see the section referred to in the text.

The following topics are included in this chapter:

- Machine switch-on
- Programming the first part
- Graphically testing the first part
- Setting up tools
- Workpiece setup
- Running the first program

1.2 Machine switch-on

Acknowledging the power interruption and moving to the reference points

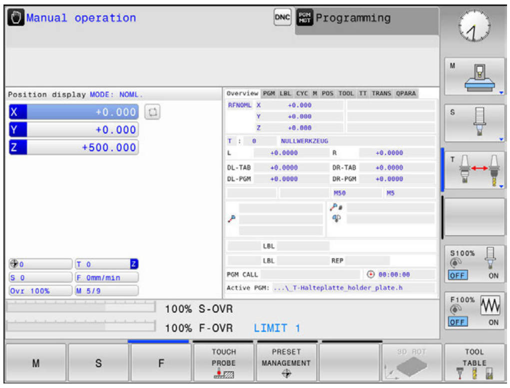
⚠DANGER

Caution: Danger for the operator!

Machines and machine components always present mechanical hazards. Electric, magnetic or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- ▶ Read and follow the machine manual
- ▶ Read and follow the safety precautions and safety symbols
- ▶ Use the safety devices

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



- ▶ Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control will then display the "Power interrupted" message in the screen header.

CE

- ▶ Press the **CE** key
- > The control compiles the PLC program.

I

- ▶ Switch on the machine control voltage
- > The control checks operation of the emergency stop circuit and goes into Reference Run mode.

- ▶ Cross the reference point manually in the prescribed sequence: For each axis press the **START** key. If you have absolute linear and angle encoders on your machine there is no need for a reference run
- > The control is now ready for operation in the **Manual operation** mode.

Further information on this topic

- Approaching reference points
  Further information: "Switch-on", page 376
- Operating modes
  Further information: "Programming", page 83

1.3 Programming the first part

Selecting the correct operating mode

You can write programs only in **Programming** mode:

- ▶ Press the operating mode key
- > The control switches to the **Programming** mode of operation.

Further information on this topic

- Operating modes
  Further information: "Programming", page 83

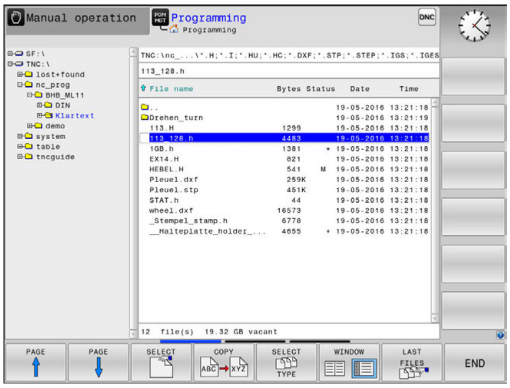
The most important control keys

Key	Functions for conversational guidance
ENT	Confirm entry and activate the next dialog prompt
NO ENT	Ignore the dialog question
END	End the dialog immediately
DEL	Abort dialog, discard entries
	Soft keys on the screen with which you select functions appropriate to the active operating state

Further information on this topic

- Writing and editing programs
  Further information: "Editing an NC program", page 119
- Overview of keys
  Further information: "Controls and displays", page 2

Opening a new program/file management



PGM MGT

- ▶ Press the **PGM MGT** key
- > The control opens the file manager.

The file management of the control is arranged much like the file management on a PC with Windows Explorer. The file management enables you to manage data in the control's internal memory.

- ▶ Use the arrow keys to select the folder in which you want to open the new file

GOTO

- ▶ Press the **GOTO** key
- > The control opens a keyboard in the pop-up window.
- ▶ Enter any desired file name with the extension **.H**

ENT

- ▶ Press the **ENT** key
- > The control asks you for the unit of measure for the new program.

MM

- ▶ Select the unit of measure: Press the **MM** or **INCH** soft key

The control automatically generates the first and last blocks of the program. Afterwards you can no longer change these blocks.

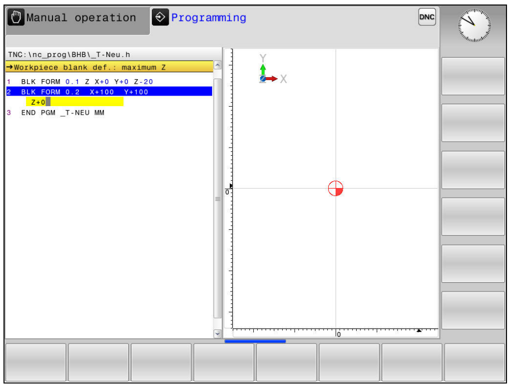
Further information on this topic

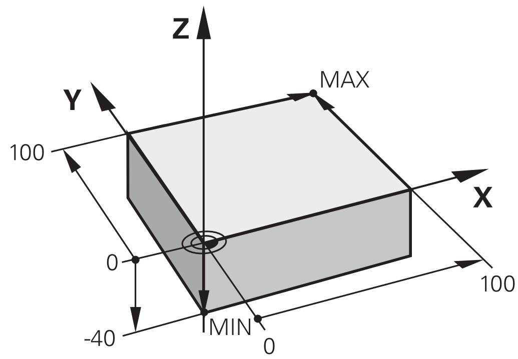
- File management
  Further information: "Working with the file manager", page 128
- Creating a new program
  Further information: "Creating and writing programs", page 112

Defining a workpiece blank

After you have created a new program you can define a workpiece blank. For example, define a cuboid by entering the MIN and MAX points, each with reference to the selected preset.

After you have selected the desired blank form via soft key, the control automatically initiates the workpiece blank definition and asks for the required data:





- **Working plane in graphic: XY?**: Enter the active spindle axis. Z is saved as default setting. Accept with the **ENT** key
- **Workpiece blank def.: Minimum X**: Enter the smallest X coordinate of the workpiece blank with respect to the preset, e.g. 0, confirm with the **ENT** key
- **Workpiece blank def.: Minimum Y**: Enter the smallest Y coordinate of the workpiece blank with respect to the preset, e.g. 0, confirm with the **ENT** key
- **Workpiece blank def.: Minimum Z**: Enter the smallest Z coordinate of the workpiece blank with respect to the preset, e.g. -40, confirm with the **ENT** key
- **Workpiece blank def.: Maximum X**: Enter the largest X coordinate of the workpiece blank with respect to the preset, e.g. 100, confirm with the **ENT** key
- **Workpiece blank def.: Maximum Y**: Enter the largest Y coordinate of the workpiece blank with respect to the preset, e.g. 100, confirm with the **ENT** key
- **Workpiece blank def.: Maximum Z**: Enter the largest Z coordinate of the workpiece blank with respect to the preset, e.g. 0, confirm with the **ENT** key

> The control ends the dialog.

Example

```
0 BEGIN PGM NEW MM
1 BLK FORM 0.1  Z X+0  Y+0 Z-40
2 BLK FORM 0.2  X+100 Y+100  Z+0
3 END PGM  NEW MM
```

Further information on this topic

- Define workpiece blank
  Further information: "Creating a new NC program", page 114

Program layout

NC programs should be arranged consistently in a similar manner. This makes it easier to find your place, accelerates programming and reduces errors.

Recommended program layout for simple, conventional contour machining

Example

```
0 BEGIN PGM  BSPCONT MM
1 BLK FORM 0.1  Z X...  Y... Z...
2 BLK FORM 0.2  X... Y...  Z...
3 TOOL CALL  5 Z  S5000
4 Z+250 R0  FMAX
5 X...   R0 FMAX
6 Z+10 R0  F3000 M13
7 X... R- F500
...
16 X...  R0 FMAX
17 Z+250 R0  FMAX M2
18 END PGM  BSPCONT MM
```

1 Call tool, define tool axis
2 Retract the tool
3 Pre-position the tool in the working plane near the contour starting point
4 In the tool axis, position the tool above the workpiece, or preposition immediately to workpiece depth. If required, switch on the spindle/coolant
5 Contour approach
6 Contour machining
7 Contour departure
8 Retract the tool, end program

Further information on this topic

- Contour programming
  Further information: "Structure blocks in NC program", page 212

Recommended program layout for simple cycle programs

Example

```
0 BEGIN PGM  BSBCYC MM
1 BLK FORM 0.1  Z X...  Y... Z...
2 BLK FORM 0.2  X... Y...  Z...
3 TOOL CALL  5 Z  S5000
4 Z+250 R0  FMAX
5 PATTERN DEF  POS1( X...  Y... Z...  ) ...
6 CYCL DEF...
7 CYCL CALL PAT  FMAX M13
8 Z+250 R0  FMAX M2
9 END PGM  BSBCYC MM
```

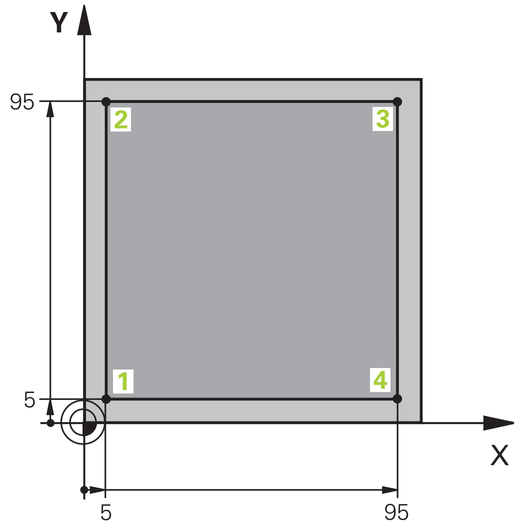
1 Call tool, define tool axis
2 Retract the tool
3 Define the machining positions
4 Define the fixed cycle
5 Call the cycle, switch on the spindle/coolant
6 Retract the tool, end program

Further information on this topic

- Cycle programming
  Further information: "Fundamentals / Overviews", page 499

Programming a simple contour

The contour shown to the right is to be milled once to a depth of 5 mm. You have already defined the workpiece blank. After you have initiated a dialog through a function key, enter all the data requested by the control in the screen header.



TOOL CALL

- Call the tool: Enter the tool data. Confirm the entry in each case with the **ENT** key, do not forget the tool axis

Z

- Retract tool: Press the orange axis key and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- Confirm **Tool radius comp: R+/R-/no comp?** with the **ENT** key: Do not activate radius compensation
- **Vorschub F=?** confirm with the **ENT** key: Rapid traverse **(FMAX)**
- Confirm **Miscellaneous function M?** with the **END** key
- > The control stores the entered positioning block.

X

- Preposition the tool in the working plane: Press the orange axis key X and enter the value for the position to be approached, e.g. –20
- Confirm **Tool radius comp: R+/R-/no comp?** with the **ENT** key: Do not activate radius compensation
- **Vorschub F=?** confirm with the **ENT** key: Rapid traverse **(FMAX)**
- Confirm **Miscellaneous function M?** with the **END** key
- > The control stores the entered positioning block.

Y

- Press the orange axis key Y and enter the value for the position to be approached, e.g. –20. Press the **ENT** key
- Confirm **Tool radius comp: R+/R-/no comp?** with the **ENT** key: Do not activate radius compensation
- **Vorschub F=?** confirm with the **ENT** key: Rapid traverse **(FMAX)**
- Confirm **Miscellaneous function M?** with the **END** key
- > The control stores the entered positioning block.

Z

- Move tool to working depth: Press the orange axis key Z and enter the value for the position to be approached, e.g. –5. Press the **ENT** key
- Confirm **Tool radius comp: R+/R-/no comp?** with the **ENT** key: Do not activate radius compensation
- **Feed rate F=?** Enter the positioning feed rate, e.g. 3000 mm/min, confirm with the **ENT** key
- **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**, and confirm with the **END** key
- > The control stores the entered positioning block.

X

- Approach contour point **1**: Press the orange X axis key and enter the value 5 for the position to be approached
- **Tool radius comp: R+/R-/no comp?** Press the R– soft key: The traverse path is decreased by the tool radius
- **Feed rate F=?** Enter the machining feed rate, e.g. 700 mm/min, save your entry with the **END** key

Y

- Approach contour point **2**: Press the orange Y axis key and enter the value 95 for the position to be approached
- **Tool radius comp: R+/R-/no comp?** Press the R + soft key: The traverse path is increased by the tool radius. Confirm your entry with the **END** key

X

- Approach contour point **3**: Press the orange X axis key and enter the value 95 for the position to be approached
- **Tool radius comp: R+/R-/no comp?** Press the R + soft key: The traverse path is increased by the tool radius. Confirm your entry with the **END** key

Y

- Approach contour point **4**: Press the orange Y axis key and enter the value 5 for the position to be approached
- **Tool radius comp: R+/R-/no comp?** Press the R + soft key: The traverse path is increased by the tool radius. Confirm your entry with the **END** key

X

- Approach contour point **1** and retract the tool: Press the orange X axis key and enter the value 0 for the position to be approached
- **Tool radius comp: R+/R-/no comp?** Press the R + soft key: The traverse path is increased by the tool radius. Confirm your entry with the **END** key

Z

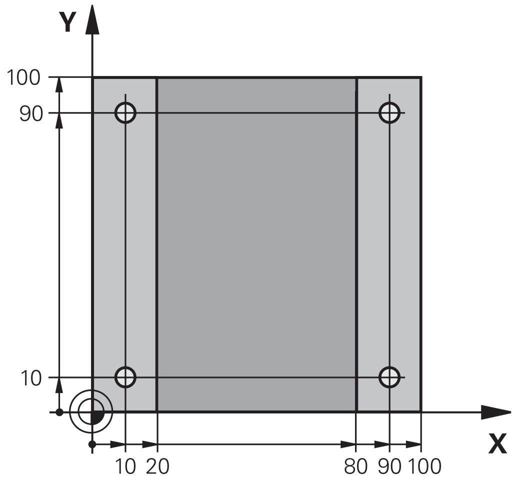
- Retract tool: Press the orange axis key Z to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- Confirm **Tool radius comp: R+/R-/no comp?** with the **ENT** key: Do not activate radius compensation
- **Vorschub F=?** confirm with the **ENT** key: Rapid traverse **(FMAX)**
- **Miscellaneous function M?** Enter **M2** to end the program, then confirm with the **END** key
- > The control stores the entered positioning block.

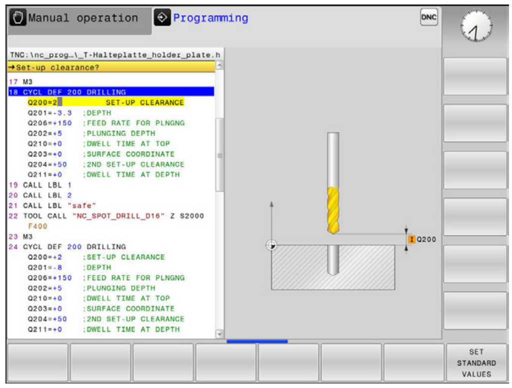
Further information on this topic

- Creating a new program
  Further information: "Creating and writing programs", page 112
- Programmable feed rates
  Further information: "Possible feed rate input", page 117
- Tool radius compensation
  Further information: "Tool radius compensation with paraxial positioning blocks", page 209
- Miscellaneous functions M
  Further information: "Miscellaneous functions for program run inspection, spindle and coolant ", page 338

Creating a cycle program

The holes (depth of 20 mm) shown in the figure at right are to be drilled with a standard drilling cycle. You have already defined the workpiece blank.





TOOL CALL

- Call the tool: Enter the tool data. Confirm the entry in each case with the **ENT** key, do not forget the tool axis

Z

- Retract tool: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- Confirm **Radius comp.: R+/R-/no comp.?** by pressing the **ENT** key: Do not activate radius compensation
- Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- **Miscellaneous function M?** Confirm with the **END** key
- > The control stores the entered positioning block.

SPEC FCT

- Call the menu for special functions: Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

- Display the functions for point machining

PATTERN DEF

- Select the pattern definition

POINT

- Select point entry: Enter the coordinates of the 4 points and confirm each with the **ENT** key. After entering the fourth point, save the block with the **END** key

CYCL DEF

- Call the cycle menu: Press the **CYCL DEF** key

DRILLING/ THREAD

- Display the drilling cycles

200

- Select standard drilling cycle 200
- > The control starts the dialog for cycle definition.
- Enter all parameters requested by the control step by step and conclude each entry with the **ENT** key
- > In the screen to the right, the control also displays a graphic showing the respective cycle parameter

CYCL CALL

- Display the menu for defining the cycle call: Press the **CYCL CALL** key

CYCLE CALL PAT

- Run the drilling cycle on the defined pattern:
- Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**, and confirm with the **END** key
- > The control stores the entered positioning block.

Z

- Enter Retract tool: Press the orange axis key **Z** and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- Confirm **Radius comp.: R+/R-/no comp.?** by pressing the **ENT** key: Do not activate radius compensation
- Confirm **Feed rate F=?** with the **ENT** key: Move at rapid traverse (**FMAX**)
- **Miscellaneous function M?** Enter **M2** to end the program, then confirm with the **END** key
- > The control stores the entered positioning block.

Example

0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S4500	Tool call
4 Z+250 R0 FMAX	Retract the tool
5 PATTERN DEF **POS1 (X+10 Y+10 Z+0)** **POS2 (X+10 Y+90 Z+0)** **POS3 (X+90 Y+90 Z+0)** **POS4 (X+90 Y+10 Z+0)**	Define the machining positions
6 CYCL DEF 200 DRILLING	Define the cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-20 ;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	
Q395=0 ;DEPTH REFERENCE	
7 CYCL CALL PAT FMAX M13	Spindle and coolant on, call the cycle
8 Z+250 R0 FMAX M2	Retract the tool, end program
9 END PGM C200 MM	

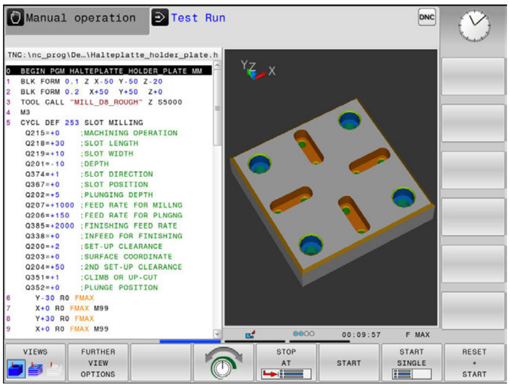
Further information on this topic

- Creating a new program
  Further information: "Creating and writing programs", page 112
- Cycle programming
  Further information: "Fundamentals / Overviews", page 499

1.4 Graphically testing the first part

Selecting the correct operating mode

You can test programs in the **Test Run** operating mode:



- ▶ Press the operating mode key
- > The control switches to the **Test Run** mode of operation.

Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", page 83
- Testing programs
  Further information: "Test run", page 444

Selecting the tool table for the test run

If you have not yet activated a tool table in **Test Run** mode, then you must carry out this step.

PGM MGT
- ▶ Press the **PGM MGT** key
- > The control opens the file manager.

SELECT TYPE
- ▶ Press the **SELECT TYPE** soft key
- > The control shows a soft-key menu for selection of the file type to be displayed.

DEFAULT
- ▶ Press the **DEFAULT** soft key
- > The control shows all saved files in the right-hand window.
- ▶ Move the cursor to the left onto the directories
- ▶ Move the cursor to the **TNC:\table** directory
- ▶ Move the cursor to the right onto the files
- ▶ Move the cursor onto the file TOOL.T (active tool table), confirm with the **ENT** key: TOOL.T contains the status **S** and is therefore active for **Test Run**

END
- ▶ Press the **END** key: Exit the file manager

Further information on this topic

- Tool management
  Further information: "Entering tool data into the table", page 188
- Testing programs
  Further information: "Test run", page 444

Choosing the program you want to test

PGM MGT

- Press the **PGM MGT** key
- > The control opens the file manager.

LAST FILES

- Press the **LAST FILES** soft key
- > The control opens a pop-up window with the most recently selected files.
- Use the arrow keys to select the program that you want to test. Load with the **ENT** key

Further information on this topic

- Program number
  Further information: "Working with the file manager", page 128

Selecting the screen layout and the view

- Press the key for selecting the screen layout
- > The control displays all available alternatives in the soft-key row.

PROGRAM + GRAPHICS

- Press the **PROGRAM + GRAPHICS** soft key
- > In the left half of the screen the control shows the program; in the right half it shows the workpiece blank.

The control features the following views:

Soft keys	Function
VIEWS	Volume view
VIEWS	Volume view and tool paths
VIEWS	Tool paths

Further information on this topic

- Graphic functions
  Further information: "Graphics ", page 432
- Performing a test run
  Further information: "Test run", page 444

Starting the test run

RESET + START

- ▶ Press the **RESET + START** soft key
- > The control resets the previously active tool data
- > The control simulates the active program up to a programmed break or to the program end
- ▶ While the simulation is running, you can use the soft keys to change views

STOP

- ▶ Press the **STOP** soft key
- > The control interrupts the test run

START

- ▶ Press the **START** soft key
- > The control resumes the test run after a break

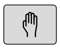
Further information on this topic

- Performing a test run
  Further information: "Test run", page 444
- Graphic functions
  Further information: "Graphics ", page 432
- Adjusting the simulation speed
  Further information: "Speed of the setting test runs", page 433

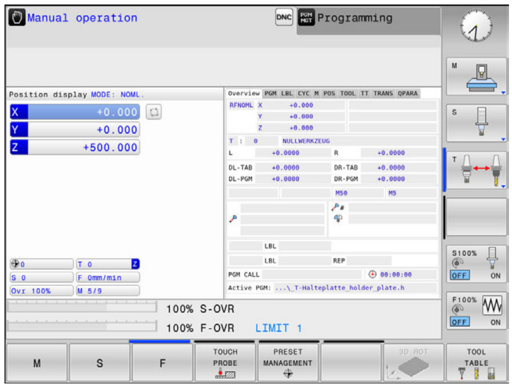
1.5 Setting up tools

Selecting the correct operating mode

Tools are set up in the **Manual operation** mode:



- ▶ Press the operating mode key
- \> The control switches to the **Manual operation** mode.



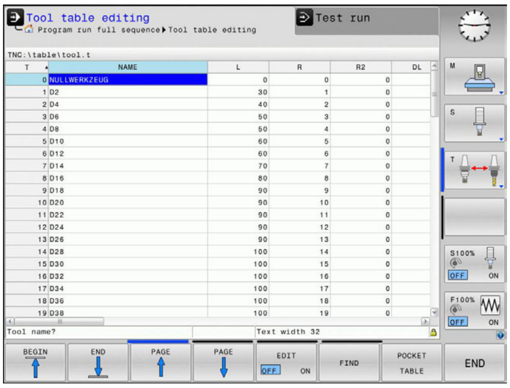
Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", page 83

Preparing and measuring tools

- ▶ Clamp the required tools in their tool holders
- ▶ When measuring with an external tool presetter: Measure the tools, note down the length and radius, or transfer them directly to the machine through a transfer program
- ▶ When measuring on the machine: Insert the tool

The tool table TOOL.T



In the TOOL.T tool table (permanently stored under **TNC:\table**), you can save tool data such as length and radius, as well as further tool-specific information that the control needs in order to execute a wide variety of functions.

To enter tool data in the tool table TOOL.T, proceed as follows:

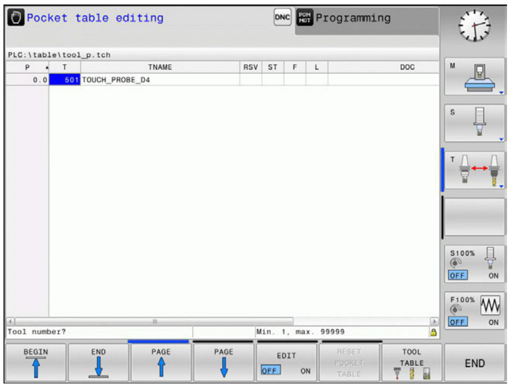
- ▶ Display the tool table
- > The control shows the tool table.

- ▶ Edit the tool table: Set the **EDIT** soft key to **ON**
- ▶ With the upward or downward arrow keys you can select the tool number that you want to edit
- ▶ With the rightward or leftward arrow keys you can select the tool data that you want to edit
- ▶ To exit the tool table, press the **END** key

Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", page 83
- Working with the tool table
  Further information: "Entering tool data into the table", page 188

The pocket table TOOL_P.TCH



> Refer to your machine manual.
> The function of the pocket table depends on the machine.

In the pocket table TOOL_P.TCH (permanently saved under **TNC:\table**) you specify which tools your tool magazine contains.

To enter data in the pocket table TOOL_P.TCH, proceed as follows:

- ▶ Display the tool table
- > The control shows the tool table.

- ▶ Display the pocket table
- > The control shows the pocket table.
- ▶ Edit the pocket table: Set the **EDIT** soft key to **ON**
- ▶ With the upward or downward arrow keys you can select the pocket number that you want to edit
- ▶ With the rightward or leftward arrow keys you can select the data that you want to edit
- ▶ To leave the pocket table, press the **END** key

Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", page 83
- Working with the pocket table
  Further information: "Pocket table for tool changer", page 199

1.6 Workpiece setup

Selecting the correct operating mode

Workpieces are set up in the **Manual operation** or **Electronic handwheel** mode

- ▶ Press the operating mode key
- > The control switches to the **Manual operation** mode.

Further information on this topic

- The operating mode **Manual operation**
  Further information: "Moving the machine axes", page 380

Clamping the workpiece

Mount the workpiece with a fixture on the machine table so that it is fixed with its edges parallel to the machine axes.

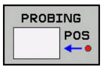
Further information on this topic

- Presetting with a 3-D touch probe
  Further information: "Presetting with a 3-D touch probe (option number 17)", page 418
- Presetting without 3-D touch probe
  Further information: "Presetting without a 3-D touch probe", page 403

Presetting with a 3-D touch probe (option number 17)

- Insert a 3-D touch probe: In the **Positioning w/ Manual Data Input** mode, run a **TOOL CALL** block containing the tool axis and then return to the **Manual operation** mode

- Press the **TOUCH PROBE** soft key
- > The control displays the available functions in the soft-key row.



- Select the function for setting a preset, e.g. press the **PROBING POS** soft key
- Use the axis direction keys to position the touch probe near the first touch point on the first workpiece edge
- Select the probing direction via soft key
- Press the **NC start** key
- > The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.
- > The control then displays the coordinates of the measured position.
- > The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.
- > The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.
- > The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

- To set to 0: Press the **SET PRESET** soft key
- Press the **END** soft key to close the menu

- Repeat this procedure for all axes, in which you want to set a preset

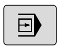
Further information on this topic

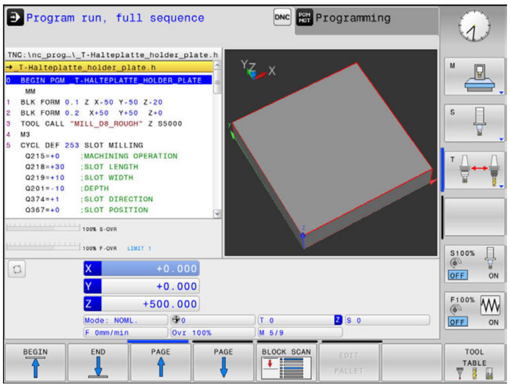
- Presetting
  Further information: "Presetting with a 3-D touch probe (option number 17)", page 418

1.7 Running the first program

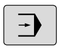
Selecting the correct operating mode

You can run programs either in the **Program run, single block** or the **Program run, full sequence** mode:





- ▶ Press the operating mode key
- > The control switches to the operating mode **Program run, single block**, and executes the NC program block by block.
- ▶ You have to confirm each block with the **NC start key**



- ▶ Press the operating mode key
- > The control switches to the operating mode **Program run, full sequence**, and executes the program after NC start up to a program interruption or to the end of the program

Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", page 83
- Executing a program
  Further information: "Program run", page 447

Choosing the program you want to run

- ▶ Press the **PGM MGT** key
- > The control opens the file manager.

- ▶ Press the **LAST FILES** soft key
- > The control opens a pop-up window with the most recently selected files.
- ▶ Use the arrow keys if required to select the program you want to run. Load with the **ENT** key

Further information on this topic

- File management
  Further information: "Working with the file manager", page 128

Starting the program

- ▶ Press the **NC start** key
- > The control runs the active program.

Further information on this topic

- Executing a program
  Further information: "Program run", page 447

2

Introduction

2.1 The TNC 128

The TNC 128 is a workshop-oriented straight-cut control that enables you to program conventional machining operations right at the machine in the easy-to-use Klartext conversational language. It is designed for milling, drilling and boring machines with up to 3 axes. You can also change the angular position of the spindle under program control.

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

HEIDENHAIN Klartext

HEIDENHAIN Klartext, the dialog-guided programming language for workshops, is an especially easy method of writing programs. Programming graphics illustrate the individual machining steps for programming the contour. Workpiece machining can be graphically simulated either during a test run or during a program run.

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

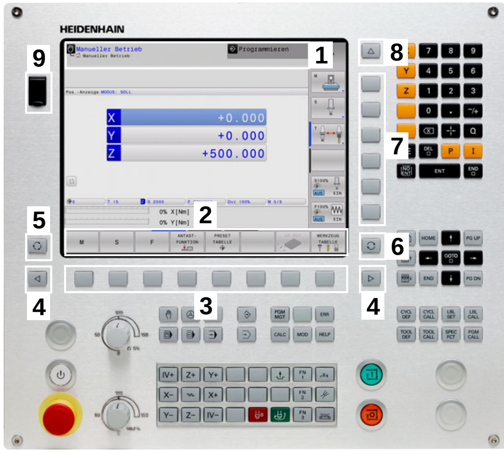
Compatibility

Machining programs created on the HEIDENHAIN TNC 124 straight cut control may not always run on the TNC 128. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.

2.2 Visual display unit and operating panel

Display screen

The control is shipped with a 12.1-inch TFT flat-panel display.



1 Header

When the control is on, the screen displays the selected operating modes in the header: The machine operating mode at left and the programming mode at right. The currently active mode is displayed in the larger field of the header, where the dialog prompts and messages also appear.

2 Soft keys

In the footer the control indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The thin bars immediately above the soft-key row indicate the number of soft-key rows that can be called with the keys to the right and left that are used to switch the soft keys. The bar representing the active soft-key row is blue

3 Soft-key selection keys

4 Keys for switching the soft keys

5 Setting the screen layout

6 Key for switchover between machine operating modes, programming modes, and a third desktop

7 Soft-key selection keys for machine tool builders

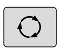
8 Keys for switching the soft keys for machine tool builders

9 USB connection

Setting screen layout

The screen layout is user-selectable. In the **Programming** mode, for example, you can have the control 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.

Set up screen layout:



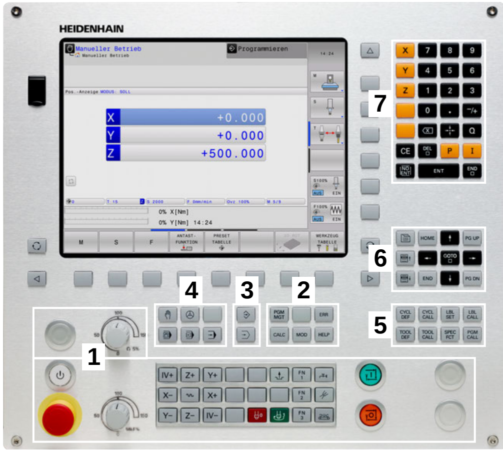
- Press the **screen layout** key: The soft-key row shows the available layout options
  Further information: "Modes of operation", page 83

PROGRAM + GRAPHICS

- Select the desired screen layout with a soft key

Control panel

The TNC 128 is delivered with an integrated operating panel.



1 Machine operating panel
More information: Machine manual

2
- File management
- Calculator
- MOD function
- HELP function
- Show error messages

3 Programming modes

4 Machine operating modes

5 Initiating programming dialogs

6 Navigation keys and **GOTO** jump command

7 Numerical input, axis selection and programming of positioning blocks

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

Refer to your machine manual.

Some machine tool builders do not use the standard HEIDENHAIN operating panel.

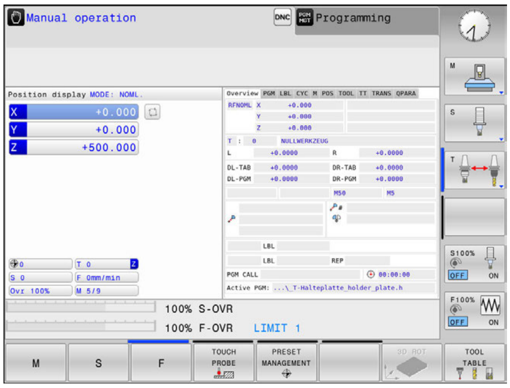
External keys, e.g.**NC START** or **NC STOP**, are described in your machine manual.

2.3 Modes of operation

Manual Operation and El. 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 and set the presets.

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

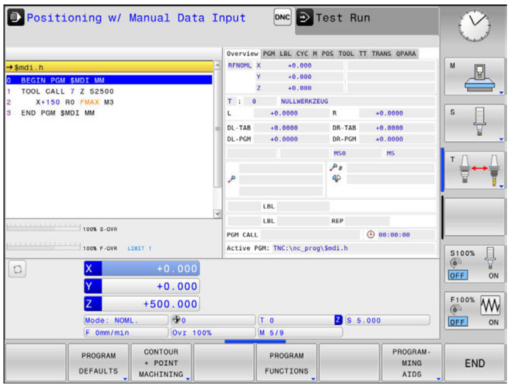


Soft keys for the screen layout (select as described above)

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

Positioning with Manual Data Input

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

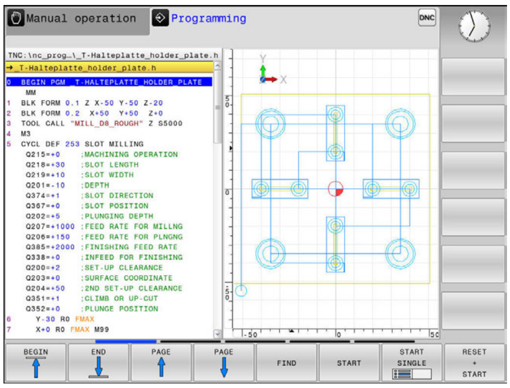


Soft keys for selecting the screen layout

Soft key	Window
PGM	Program
PROGRAM + STATUS	Left: program, right: status display

Programming

In this mode of operation you create NC programs. The various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

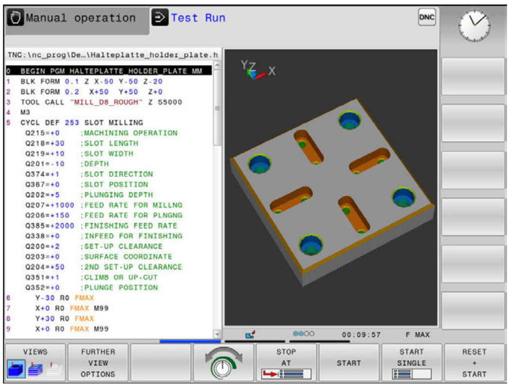


Soft keys for selecting the screen layout

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

Test Run

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



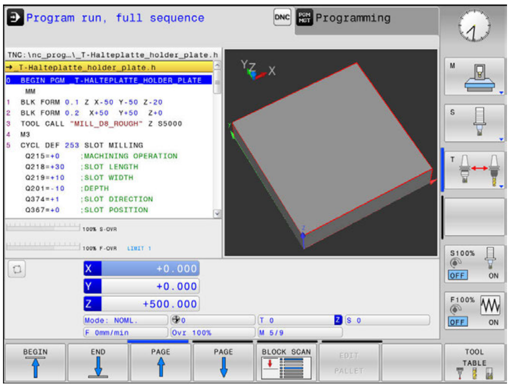
Soft keys for selecting the screen layout

Soft key	Window
PGM	Program
PROGRAM + STATUS	Left: program, right: status display
PROGRAM + GRAPHICS	Left: program, right: graphics
GRAPHICS	Graphic

Program Run, Full Sequence and Program Run, Single Block

In the **Program Run Full Sequence** mode, the control executes a 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, you execute each block separately by pressing the **NC start** key. With point pattern cycles and **CYCL CALL PAT** the controls stops after each point.

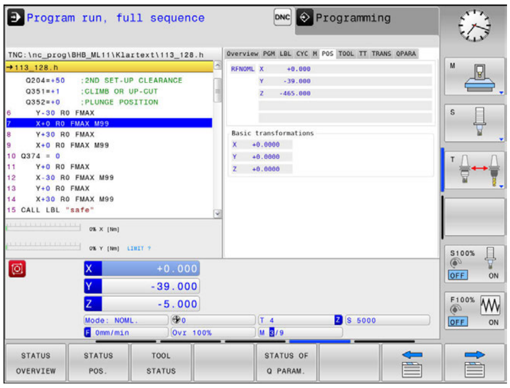


Soft keys for selecting the screen layout

Soft key	Window
PGM	Program
PROGRAM + SECTS	Left: program, right: structure
PROGRAM + STATUS	Left: program, right: status display
PROGRAM + GRAPHICS	Left: program, right: graphics
GRAPHICS	Graphic

2.4 Status displays

General status display



The general status display in the lower part of the screen informs you of the current state of the machine.

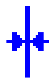
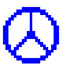
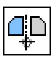
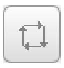
It is displayed automatically in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Positioning w/ Manual Data Input**

> If the **GRAPHICS** screen layout is selected the status display is not shown.

In the **Manual operation** and **Electronic handwheel** modes the status display appears in the large window.

Information in the status display

Icon	Meaning
ACTL	Position display mode, e.g. actual or nominal coordinates of the current position
X Y Z	Machine axes; the control 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
	Axis is clamped
	Axis can be moved with the handwheel
	Axes are mirrored and moved
	No program selected, program reselected, program aborted via internal stop or program terminated In this condition the control has no modally effective program information (i.e. the contextual reference), so that all handling is possible, e.g. cursor movements or modification of Q parameters.
	Program started, execution runs For safety reasons, the control permits no handling in this condition
	Program stopped, e.g. in operating mode **Program run, full sequence** after pressing the **NC stop** key For safety reasons, the control permits no handling in this condition

Icon	Meaning
	Program interrupted, e.g. in operating mode **Positioning w/ Manual Data Input** following the error-free execution of an NC block In this condition the control permits various handling, e.g. cursor movements or the modification of Q parameters. With this handling the control may lose the modally effective program information (i.e. the contextual reference). Loss of this contextual reference may cause undesired tool positions! **Further information:** "Programming and executing simple machining operations", page 426 and "Program-controlled interruptions", page 449
	Program aborted or terminated
	Pulsing spindle speed function is active

> The order of icons can be changed with the optional machine parameter **iconPrioList** (no. 100813). The control-in-operation symbol is always visible and cannot be configured.

Additional status displays

The additional status displays contain detailed information on the program run. This can be called in all operating modes except for the **Programming** mode.

To switch on the additional status display

- ▶ Call the soft key row for screen layout

PROGRAM + STATUS

- ▶ Select the layout option for the additional status display
- > In the right half of the screen, the control shows the **Overview** status form.

To select an additional status display

- ▶ Toggle through the soft key rows until the **STATUS** soft keys appear

STATUS POS.

- ▶ Either select the additional status display directly with the soft key, e.g. positions and coordinates; or
- ▶ use the switch-over soft keys to select the desired view

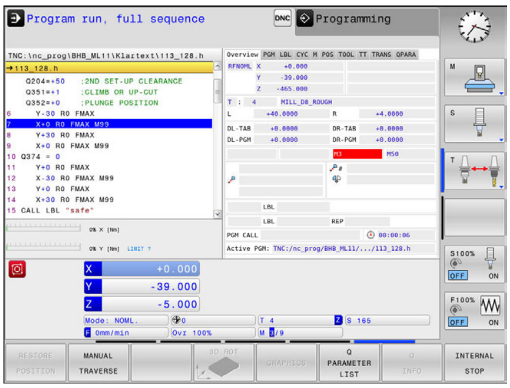
Select the status displays described below as follows:

- directly with the corresponding soft key
- via the switchover soft keys
- or by using the **next tab** key

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

Overview

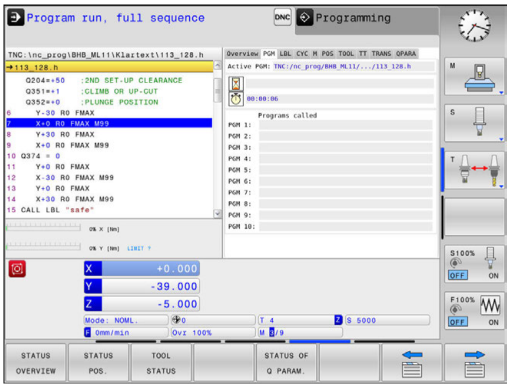
The **Overview** status form is displayed by the control following switch-on if you selected the screen layout **PROGRAM + STATUS** (or **POSITION + STATUS**). The overview form contains a summary of the most important status information, which you can also find on the various detail forms.



Soft key	Meaning
STATUS OVERVIEW	Position display
	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program section repeat
	Program called with **PGM CALL**
	Current machining time
	Name and path of the active main program

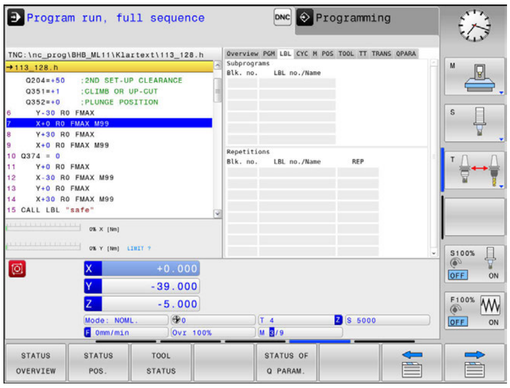
General program information (PGM tab)

Soft key	Meaning
No direct selection possible	Name and path of the active main program
	Actual/nominal value counter
	Dwell time counter
	Current machining time
	Active programs



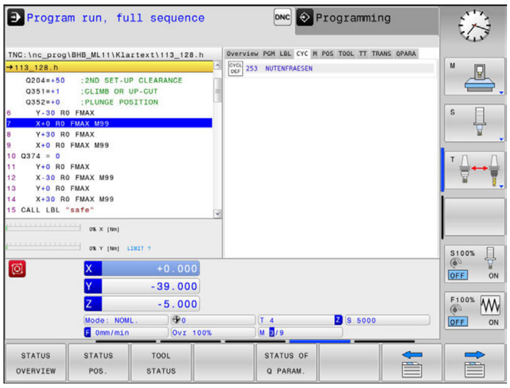
Program section repeats and subprograms (LBL tab)

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 subprograms with block number in which the subprogram was called and the label number that was called



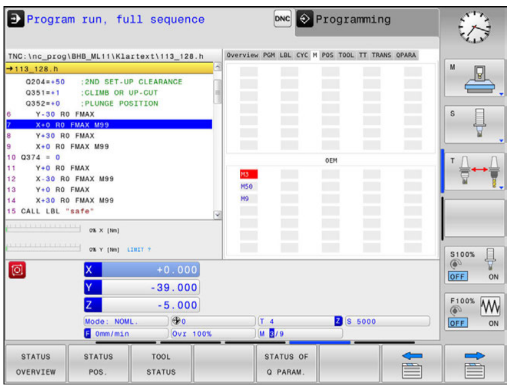
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active fixed cycle



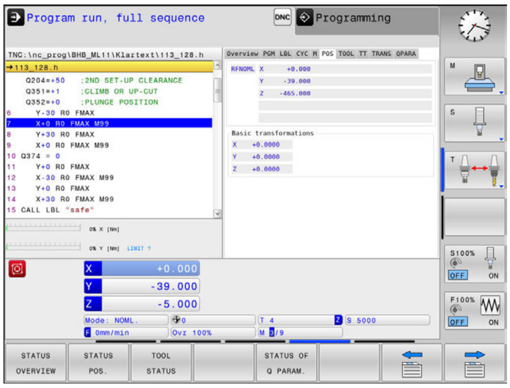
Active miscellaneous functions M (M tab)

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



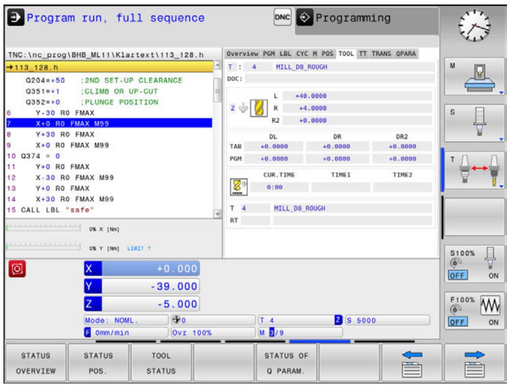
Positions and coordinates (POS tab)

Soft key	Meaning
STATUS POS.	Type of position display, e.g. actual position



Information on tools (TOOL tab)

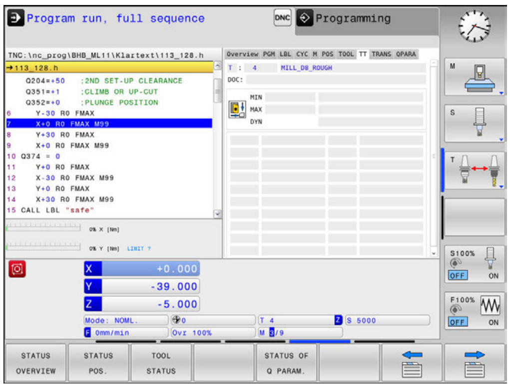
Soft key	Meaning
TOOL STATUS	Display of active tool: ■ T: Tool number and tool name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool length and tool radii
	Oversizes (delta values) from the tool table (TAB) and the **TOOL CALL** (PGM)
	Tool life, maximum tool life (TIME 1) and maximum tool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



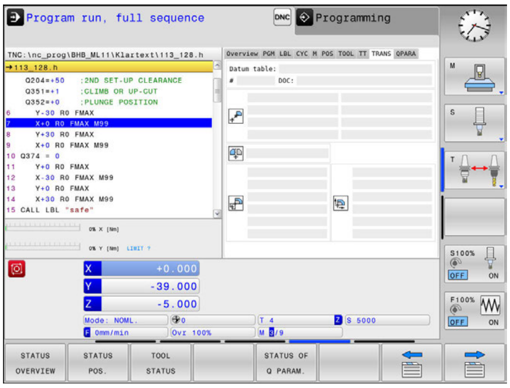
Tool measurement (TT tab)

> The control displays this tab only if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Active tool
	Measured values from tool measurement



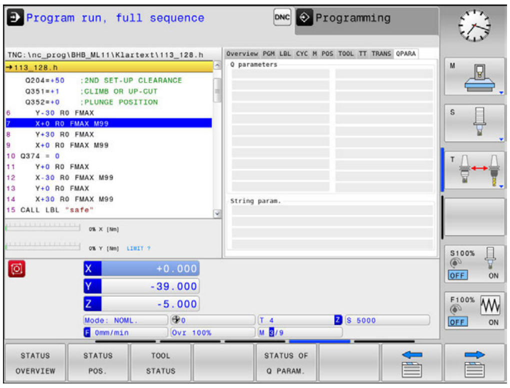
Coordinate transformations (TRANS tab)



Soft key	Meaning
No direct selection possible	Name of the active datum table
	Active datum number (**#**), comment from the active line of the active datum number (**DOC**) from Cycle 7
	Active datum shift (Cycle 7); the control displays an active datum shift in up to 3 (5) axes
	Mirrored axes (Cycle 8)
	Active scaling factor/factors (Cycle 11 / 26); the control displays an active scaling factor in up to 6 axes
	Scaling datum

Cycles for coordinate transformation
Further information: "Cycles: Coordinate Transformations", page 593

Displaying Q parameters (QPARA tab)



Soft key	Meaning
STATUS OF Q PARAM.	Display the current values of the defined Q parameters
	Display the character strings of the defined string parameters

Press the **Q PARAMETER LIST** soft key. The control opens a pop-up window. For each parameter type (Q, QL, QR, QS) define the parameter numbers you wish to control. Separate single Q parameters with a comma, and connect sequential Q parameters with a hyphen, e.g. 1,3,200-208. The input range per parameter type is 132 characters.

The display in the **QPARA** tab always contains eight decimal places. The result of Q1 = COS 89.999 is shown by the control as 0.00001745, for example. Very large and very small values are displayed by the control in exponential notation. The result of Q1 = COS 89.999 * 0.001 is shown by the control as +1.74532925e-08, whereby e-08 corresponds to the factor of 10^{-8}.

2.5 Window manager

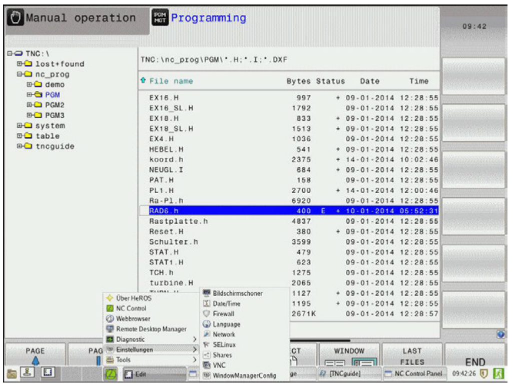
> Refer to your machine manual.
> The machine tool builder determines the scope of function and behavior of the window manager.

The control features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces. The following functions are possible with the window manager:

- Display a task bar for switching between various applications (user interfaces)
- Manage an additional desktop, on which special applications from your machine manufacturer can run
- Control the focus between NC software applications and those of the machine manufacturer
- You can change the size and position of pop-up windows. It is also possible to close, minimize and restore pop-up windows

> The control shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

Overview of taskbar



In the task bar you can choose different workspaces by mouse click.

The control provides the following workspaces:

- Workspace 1: Active operating mode
- Workspace 2: Active programming mode
- Workspace 3: or applications of the machine tool builder (optionally available)
- Workspace 4: applications of the machine tool builder (optionally available)

In addition, you can also select other applications from the task bar which you have started in parallel to the control software, e.g. the **TNCguide**.

> You can randomly move all open applications to the right of the green HEIDENHAIN symbol between the workspaces by pressing and holding the left mouse button.

Click the green HEIDENHAIN symbol to open a menu in which you can get information, make settings or start applications.

The following functions are available:

- **About HEROS**: Open information about the operating system of the control
- **NC Control**: Start and stop the control software (for diagnostic purposes only)
- **Web Browser**: Start the web browser
- **Diagnostic**: Diagnostic applications
  - **GSmartControl**: Available only to authorized specialists
  - **HE Logging**: Define settings for internal diagnostic files
  - **HE Menu**: Available only to authorized specialists
  - **perf2**: Check processor load and process load
  - **Portscan**: Test active connections
    Further information: "Portscan", page 94
  - **Portscan OEM**: Available only to authorized specialists
  - **RemoteService**: Start and stop remote maintenance
    Further information: "Remote Service", page 95
  - **Terminal**: Enter and execute console commands
- **Settings**: Operating system settings
  - **Date/Time**: Set date and time
  - **Language/Keyboards**: Select system dialog language and keyboard version—the control overwrites the setting of the system dialog language when starting with the language setting of the machine parameter **CfgDisplayLanguage** (no. 101300)
  - **Network**: Define network settings
  - **Printer**: Configure and manage printer
    Further information: "Printer", page 97
  - **Screensaver**: Define screensaver

  - **SELinux**: Define safety software for Linux-based operating systems
  - **Shares**: Connect and manage external network drives
  - **VNC**: Define the setting for external software accessing the control for e.g. maintenance work (**V**irtual **N**etwork **C**omputing)
    Further information: "VNC", page 100
  - **WindowManagerConfig**: Available only to authorized specialists
  - **Firewall**: Configure the firewall
    Further information: "Firewall", page 491
  - **HePacketManager**: Available only to authorized specialists
  - **HePacketManager Custom**: Available only to authorized specialists
- **Tools**: File applications
  - **Document Viewer**: Display and print files, e.g. PDF files
  - **File Manager**: Available only to authorized specialists
  - **Geeqie**: Open, manage, and print graphics
  - **Gnumeric**: Open, edit, and print tables
  - **Keypad**: Open virtual keyboard
  - **Leafpad**: Open and edit text files
  - **NC/PLC Backup**: Create backup file
    Further information: "Backup and restore", page 102
  - **NC/PLC Restore**: Restore backup file
    Further information: "Backup and restore", page 102
  - **Ristretto**: Open graphics
  - **Screenshot**: Create screenshots
  - **TNCguide**: Call up help system
  - **Xarchiver**: Extract or compress directories
  - **Applications**: Supplementary applications
    - **Orage Calender**: Open calendar
    - **Real VNC viewer**: Define the setting for external software accessing the control for e.g. maintenance work (Virtual Network Computing)

> The applications available under tools can be started directly by selecting the corresponding file type in the file management of the control
> **Further information:** "Additional tools for management of external file types", page 141

Portscan

The PortScan function enables the cyclic or manual searching for all open, incoming TCP and UDP list ports on the system. All ports found are compared with whitelists. If the control finds a non-listed port it shows a corresponding pop-up window.

The HeROS **Diagnostic** menu contains the **Portscan** and **Portscan OEM** applications for this purpose. **Portscan OEM** is only executable after entering the machine manufacturer password.

The **Portscan** function searches for all open, incoming TCP and UDP ports on the system and compares them to four whitelists stored in the system:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with machine manufacturer-specific functions, e.g. for Python and DNC applications: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

For each entry, the whitelist specifies the type of port (TCP/UDP), the port number, the providing program, and optional comments. If the automatic port scan function is active, only ports listed in the whitelists can be open. Non-listed ports trigger a notification window.

The result of the scan is saved to a log file (LOG:/portscan/scanlog and LOG:/portscan/scanlogevil), and if new ports are found that are not listed in one of the whitelists these are displayed.

Manually starting Portscan

Proceed as follows to manually start the Portscan:

- ▶ Taskbar at the bottom edge of the screen
  Further information: "Window manager", page 91
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **Portscan** menu item
- > The control opens the **HeRos Portscan** pop-up window.
- ▶ Press the **Start** key

Cyclically starting Portscan

Proceed as follows to automatically start the Portscan cyclically:

- ▶ Taskbar at the bottom edge of the screen
  Further information: "Window manager", page 91
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **Portscan** menu item
- > The control opens the **HeRos Portscan** pop-up window.
- ▶ Press the **Automatic update on** key
- ▶ Set the time interval with the slider

Remote Service

Together with the Remote Service Setup Tool, the TeleService from HEIDENHAIN enables encrypted end-to-end connections to be established between a service computer and the machine tool.

To enable the HEIDENHAIN control to communicate with the HEIDENHAIN server it must be connected to the internet.

Further information: "Configuring the control", page 486

In its basic state the firewall of the control blocks all incoming and outgoing connections. For this reason the firewall must be deactivated for the duration of the service session.

Setting up the control

To set up the control, proceed as follows:

▶ Taskbar at the bottom edge of the screen
Further information: "Window manager", page 91

▶ Press the green HEIDENHAIN button to open the JH menu

▶ Select the **Settings** menu item

▶ Select the **Firewall** menu item

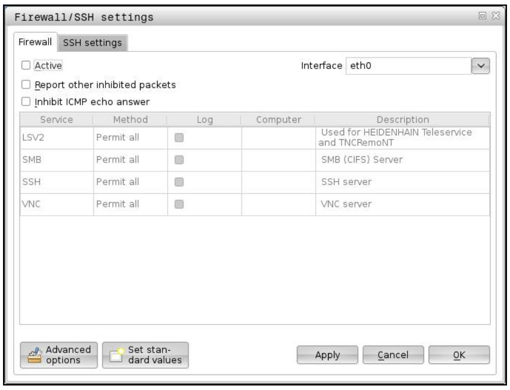
> The control displays the **Firewall/SSH settings** dialog

▶ Deactivate the firewall by removing the **Active** option in the **Firewall** tab.

▶ Press the **Apply** button to save the settings

▶ Press the **OK** button

> The firewall is disabled.

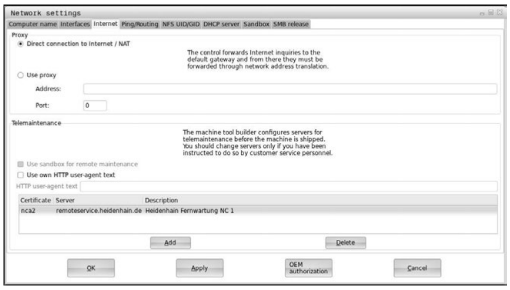


ℹ	Do not forget to activate the firewall again after the end of the service session.

Automatic installation of a session certificate

With an NC software installation a temporary certificate is automatically installed on the control. An installation, also in the form of an update, may only be carried out by a service technician from the machine tool builder.

Manual installation of a session certificate



A new certificate must be installed if no valid session certificate is installed on the control. Clarify which certificate is needed with your service employee. He will then provide you with a valid certificate file if necessary.

To install the certificate on the control proceed as follows:

- Taskbar at the bottom edge of the screen
  Further information: "Window manager", page 91
- Press the green HEIDENHAIN button to open the JH menu
- Select the **Settings** menu item
- Select the **Network** menu item
- > The control displays the **Network settings** dialog
- Select the **Internet** tab. The settings in the **Remote maintenance** field are configured by the machine tool builder.
- Press the **Add** key and select the file from the menu
- Press the **Open** key
- > The certificate is opened.
- Press the **OK** soft key
- It may be necessary to restart the control to load the settings

Launching the service session

Proceed as follows to start the service session:

- Open the task bar at the bottom edge of the screen
- Press the green HEIDENHAIN button to open the JH menu
- Select the **Diagnostic** menu item
- Select the **RemoteService** menu item
- Enter the **Session key** of the machine tool builder

Printer

The **Printer** function in the HeROS menu enables you to add and manage printers.

Displaying the printer settings

Proceed as follows to access the printer settings:

- ▶ Open the task bar at the bottom edge of the screen
  Further information: "Window manager", page 91
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Settings** menu item
- ▶ Select the **Printer** menu item
- > The control opens the **Heros Printer Manager** pop-up window.

The name of the printer is displayed in the input field.

Soft key	Meaning
CREATE	Creates the printer named in the input field
CHANGE	Modifies the properties of the selected printer
COPY	Creates the printer named in the input field with the attributes of the selected printer This can be useful if printing both portrait and landscape formats on the same printer
DELETE	Deletes the selected printer
UP DOWN	Selects the desired printer
STATUS	Displays status information of the selected printer
PRINT TEST PAGE	Prints a test page on the selected printer

For each printer, the following attributes can be set:

Setting	Meaning
Name of the printer	The printer name can be changed here.
Connection	Here, you can select the connection type ■ USB: The USB connection can be assigned here. The name is displayed automatically. ■ Network: Enter the network name or the IP address of the target printer here. In addition, specify the port of the network printer here (default: 9100) ■ Printer not connected
Timeout	Defines the delay to printing after the last change has been made to the file to be printed in PRINTER:. This can be useful if the file to be printed is populated with data by using FN functions, e.g. during probing.
Standard printer	Select to define the standard printer in case several printers are available. Is defined automatically when creating the first printer.

Setting	Meaning
Settings for printing of text	These settings are applicable when printing text documents: ■ Paper size ■ Number of copies ■ Job name ■ Font size ■ Header ■ Print options (black and white, color, duplex)
Orientation	Portrait, landscape for all printable files
Expert options	Available only to authorized specialists

Print options:

- Copying of the file to be printed in PRINTER:
  The file to be printed is automatically forwarded to the standard printer and deleted from the directory after the print job has been executed
- Using the FN 16: F-PRINT function
  Further information: "Printing messages", page 267

List of printable files:

- Text files
- Graphic files
- PDF files

> The connected printer must be PostScript-enabled.

SELinux security software

SELinux is an extension for Linux-based operating systems. SELinux is an additional security software package based on Mandatory Access Control (MAC) and protects the system against the running of unauthorized processes or functions and therefore protects against viruses and other malware.

MAC means that every action must be explicitly permitted, otherwise it will not be executed by the control. The software is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can only be executed if the standard functions and access control of SELinux permit it.

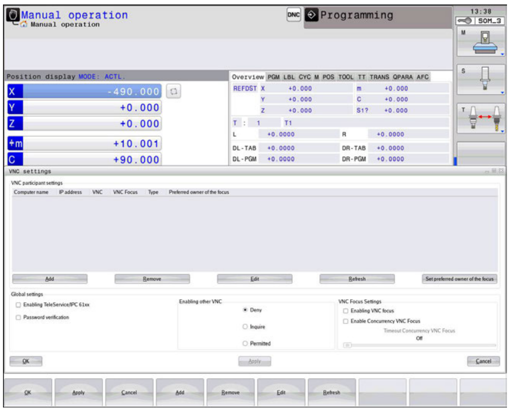
> The SELinux installation of the control has been prepared to permit running only programs installed with the HEIDENHAIN NC software. Other programs cannot be run with the standard installation.

The access control of SELinux under HEROS 5 is regulated as follows:

- The control executes only applications that are installed with the HEIDENHAIN NC software
- Files in connection with the security of the software (SELinux system files, HEROS 5 boot files, etc.) may only be changed by programs that are selected explicitly
- New files generated by other programs must never be executed
- USB data carriers cannot be deselected
- There are only two processes that are permitted to execute new files:
  - Starting a software update: A software update from HEIDENHAIN can replace or change system files
  - Starting the SELinux configuration: The configuration of SELinux is usually password-protected by your machine manufacturer; refer here to the relevant machine manual

> HEIDENHAIN recommends activating SELinux because it provides additional protection against attacks from outside.

VNC



Use the **VNC** function to configure the behavior of the various VNC clients. This includes, for example, operation via soft keys, mouse and the ASCII keyboard.

The control provides the following options:

- List of permitted clients (IP address or name)
- Password for the connection
- Additional server options
- Additional settings for assigning the focus

> Refer to your machine manual.
>
> For multiple clients or operating units, the focus assignment procedure depends on the design and the operating situation of the machine.
>
> This function must be adapted by your machine manufacturer.

Opening the VNC settings

Proceed as follows to open the VNC settings:

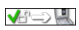
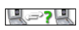
- ▶ Open the task bar at the bottom edge of the screen
  Further information: "Window manager", page 91
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Settings** menu item
- ▶ Select the **VNC** menu item
- > The control opens the **VNC Settings** pop-up window.

The control provides the following options:

- Add: Add new VNC viewer/client
- Remove: Deletes the selected client Only possible with manually entered clients.
- Edit: Edit the configuration of the selected client
- Update: Updates the display. Required with connection attempts during which the dialog is open.

VNC settings

Dialog	Option	Meaning
VNC participant settings	**Computer name:**	IP address or computer name
	VNC:	Connection of the client to the VNC viewer
	VNC Focus	The client participates in the focus assignment
	Type	■ Manual Manually entered client ■ Denied This client is not permitted to connect ■ TeleService/IPC 61xx Client via TeleService connection ■ DHCP Other computer that obtains an IP address from this computer

Dialog	Option	Meaning
Firewall warning		Warnings and information about if the VNC protocol has not been authorized for all VNC clients due to firewall settings on the control. **Further information:** "Firewall", page 491.
Global settings	**Enabling TeleService/ IPC 61xx**	Connection via TeleService/IPC 61xx is always permitted
	Password verification	The client must enter a password for verification. If this option is active, the password must be entered when the connection is established.
Enabling other VNC	**Deny**	Access generally denied to all other VNC clients.
	Inquire	During connection attempts a corresponding dialog is opened.
	Permitted	Access is generally granted to all other VNC clients.
VNC Focus Settings	**Enabling VNC focus**	Enable focus assignment for this system. Otherwise there is no central focus assignment. In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus.
	Enabling concurrency VNC focus	In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus. If concurrency focus is selected, any client can retrieve the focus at any time without having to wait for the current owner of the focus to release it.
	Timeout Concurrency VNC Focus	Time period within which the current owner of the focus can object to the focus being withdrawn or can prevent the reassignment of the focus. If a client requests the focus, a dialog in which the reassignment of focus can be refused appears on all clients' screens.
Focus symbol		Current status of VNC focus on the respective client: Focus is owned by other client. Mouse and keyboard are disabled.
		Current status of VNC focus on respective client: Focus is owned by current client. Entries can be made.
		Current status of VNC focus on the respective client: Request by the owner of the focus to give the focus to another client. Mouse and keyboard are disabled until the focus is assigned unambiguously.

If **Enable concurrency VNC focus** is selected, a pop-up window appears. This dialog makes it possible to refuse that the focus be given to the requesting client. If this does not occur, the focus changes to the requesting client after the set time limit.

Backup and restore

With the **NC/PLC Backup** and **NC/PLC Restore** functions you can back up and restore individual folders or the complete **TNC** drive. You can save the backup files locally, on a network drive, or to USB storage devices.

The backup program generates a ***. tncbck** file that can also be processed by the PC tool TNCbackup (part of TNCremo). The restore program can restore these files as well as those from existing TNCbackup programs. If a *. tncbck file is selected in the file manager of the control, the program **NC/PLC Restore** is automatically launched.

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** soft keys. Specific actions for steps are selectively displayed as soft keys.

Opening NC/PLC Backup or NC/PLC Restore

Proceed as follows to open the functions:

- Open the task bar at the bottom edge of the screen
  Further information: "Window manager", page 91
- Press the green HEIDENHAIN button to open the JH menu
- Select the **Tools** menu item
- Open the **NC/PLC Backup** or **NC/PLC Restore** menu item
- \> The control opens the pop-up window.

Backing up data

To backup data from the control, proceed as follows:

- Select **NC/PLC Backup**
- Select the type
  - Back up the **TNC** partition
  - Back up the directory tree: Select the directory for backup in the file management
  - Back up the machine configuration (for machine tool builders only)
  - Complete backup (for machine tool builders only)
  - Comment: Freely configurable comment for the backup
- Select the next step with the **FORWARD** soft key
- Stop the control if required with the **STOP NC SOFTWARE** soft key
- Define the exclusion rules
  - Use preset rules
  - Write own rules to the table
- Select the next step with the **FORWARD** soft key
- > The control generates a list of files for backing up.
- Check the list. Deselect files if necessary.
- Select the next step with the **FORWARD** soft key
- Enter the name of the backup file
- Select the storage path
- Select the next step with the **FORWARD** soft key
- > The control generates the backup file.
- Confirm with the **OK** soft key
- > The control concludes the backup process and restarts the NC software.

Restoring data

> **NOTICE**
>
> **Caution: Data may be lost!**
>
> When you restore data (Restore function), any existing data will be overwritten without a confirmation prompt. Existing data is not automatically backed up by the control before running the restore process. Power failures or other problems can interfere with the data restore process. As a consequence, data may be irreversibly damaged or deleted.
>
> - Before starting the data restore process, make a backup of the existing data

To restore the data proceed as follows:

- Select **NC/PLC Restore**
- Select the archive to be restored
- Select the next step with the **FORWARD** soft key
- > The control generates a list of files for restoring.
- Check the list. Deselect files if necessary.
- Select the next step with the **FORWARD** soft key
- Stop the control if required with the **STOP NC SOFTWARE** soft key
- Extract the archive
- > The control restores the files.
- Confirm with the **OK** soft key
- > The control restarts the NC software.

2.6 Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels

3-D touch probes

Applications for HEIDENHAIN 3-D touch probes:

- Quickly and precisely set presets
- Measure the workpiece
- Measure and inspect tools

Touch trigger probes TS 260 and KT 130

The TS 260 and KT 130 touch probes transmit the trigger signals via a cable.

HEIDENHAIN touch trigger probes feature a wear-resistant optical switch that detects the deflection of the stylus. On deflection, a trigger signal is generated, which causes the control to store the current position of the touch probe as the actual value.

Tool touch probe TT 160

The TT 160 touch probe is designed for the efficient and precise measurement and inspection of tool dimensions.

The control offers cycles that enable you to determine the tool length and radius while the spindle is rotating or stationary. The tool touch probe features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf.

A wear-resistant optical switch generates the trigger signal. With the TT 160, signal transmission is by cable.

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 510, HR 520 and HR 550FS portable handwheels.

Further information: "Moving with the electronic display handwheels", page 383

3

Fundamentals, File Management

3.1 Fundamentals

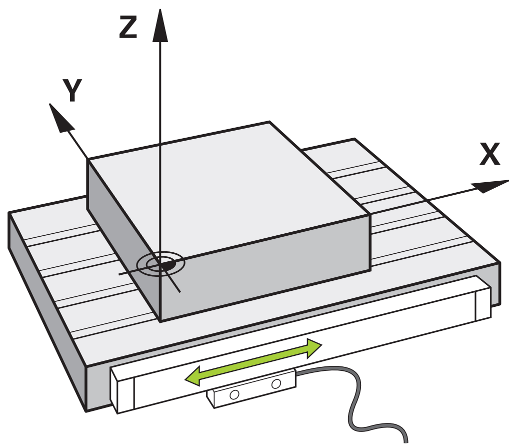
Position encoders and reference marks

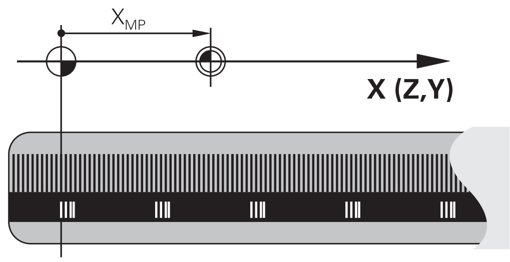
The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders, rotary tables and tilting axes with angle encoders.

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

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, 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 control when they are crossed over. This enables the control to re-establish the assignment of the displayed position to the current machine position. 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.



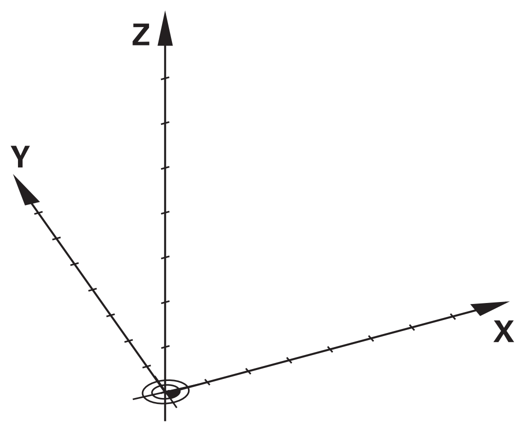


Reference system

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.

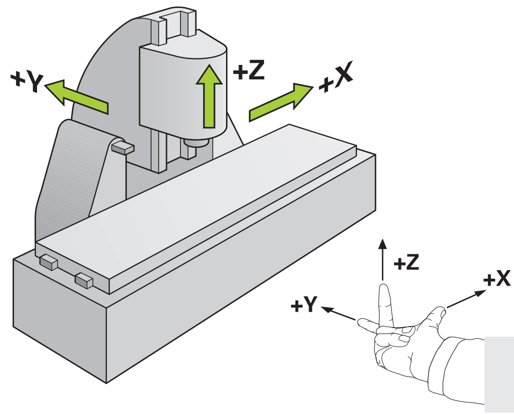
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (reference point) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.



Reference system of milling machines

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

The TNC 128 can control up to 4 axes optionally. 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 principal axes.



Designation of the axes on milling machines

The X, Y and Z axes on your milling machine are also referred to as tool axis, principal axis (1st axis) and secondary axis (2nd axis). The assignment of the tool axis is decisive for the assignment of the principal and secondary axes.

Tool axis	Principal axis	Secondary axis
X	Y	Z
Y	Z	X
Z	X	Y

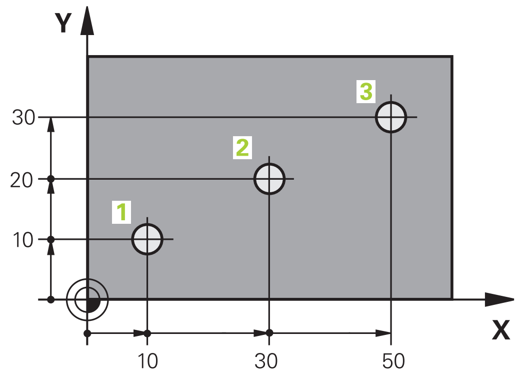
Absolute and incremental workpiece positions

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 unambiguously 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

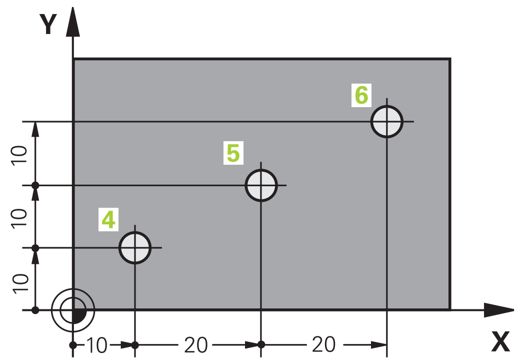


Incremental workpiece positions

Incremental coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When you write an NC 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 they are also referred to as chain dimensions.

To program a position in incremental coordinates, enter the letter **I** before the axis.

Example 2: Holes dimensioned in incremental coordinates

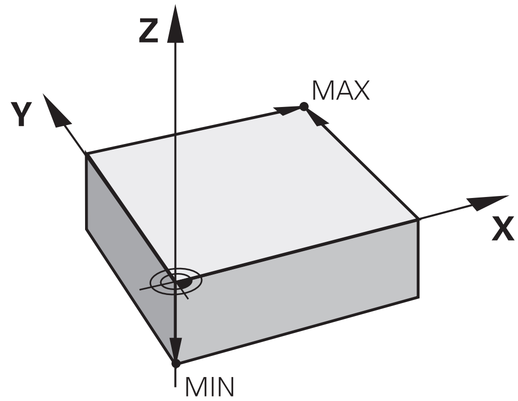


Absolute coordinates of hole 4
X = 10 mm
Y = 10 mm

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

Selecting the preset

A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute preset (datum). When setting the preset, 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 control 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 control's display and your part program.



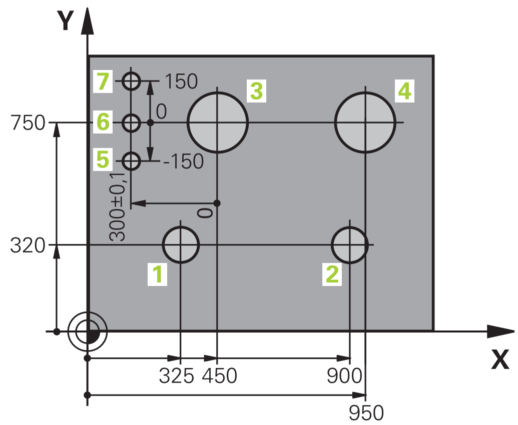
If the production drawing is dimensioned in relative presets, simply use the coordinate transformation cycles.

Further information: "DATUM SHIFT (Cycle 7)", page 595

If the production drawing is not dimensioned for NC, set the preset at a position or corner on the workpiece from which the dimensions of the remaining workpiece positions can be measured.

Example

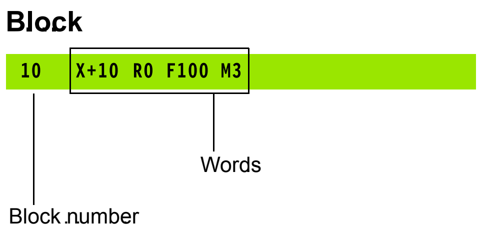
The workpiece drawing shows holes (**1** to **4**), whose dimensions are shown with respect to an absolute preset with the coordinates X=0 Y=0. The coordinates of holes **5** to **7** refer to the relative preset with the absolute coordinates X=450 Y=750. By using the **Datum shift** cycle you can shift the datum temporarily to the position X=450, Y=750 and program the holes (**5** to **7**) without further calculations.



3.2 Creating and writing programs

Structure of an NC program in HEIDENHAIN Klartext

A machining program consists of a series of NC blocks. The illustration on the right shows the elements of a block.



The control numbers the blocks of a part program in ascending order.

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

The subsequent blocks contain information on:

- The workpiece blank
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds, as well as
- Movements, cycles and other functions

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

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. There is danger of collision during the approach movement after a tool change!

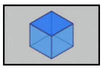
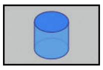
- If necessary, program an additional safe auxiliary position

Defining the blank: BLK FORM

Immediately after initiating a new program, you define an unmachined workpiece blank. If you wish to define the blank at a later stage, press the **SPEC FCT** key, the **PROGRAM DEFAULTS** soft key, and then the **BLK FORM** soft key. The control needs this definition for graphic simulation.

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

The control can depict various types of blank forms:

Soft key	Function
	Define a rectangular blank
	Define a cylindrical blank

Rectangular blank

The sides of the cuboid lie parallel to the X, Y and Z axes. This blank is defined by two of its corner points:

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

Example

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

Cylindrical blank

The cylindrical blank form is defined by the dimensions of the cylinder:

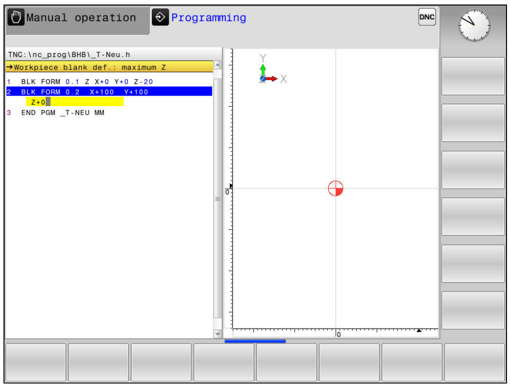
- X, Y or Z: Rotation axis
- D, R: Diameter or radius of the cylinder (with positive algebraic sign)
- L: Length of the cylinder (with positive algebraic sign)
- DIST: Shifting along the rotational axis
- DI, RI: Inside diameter or inside radius for a hollow cylinder

> The parameters **DIST** and **RI** or **DI** are optional and need not be programmed.

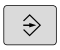
Example

0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM CYLINDER Z R50 L105 DIST+5 RI10	Spindle axis, radius, length, distance, inside radius
2 END PGM NEW MM	Program end, name, unit of measure

Creating a new NC program



You always enter an NC program in **Programming** mode. An example of program initiation:



- Operating mode: Press the **Programming** key

PGM MGT

- Press the **PGM MGT** key
- > The control opens the file manager.

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

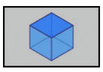
FILE NAME = NEW.H

ENT

- Enter the new program name
- Press the **ENT** key

MM

- Select the unit of measure: Press the **MM** or **INCH** soft key
- > The control switches the screen layout and initiates the dialog for defining the **BLK FORM** (workpiece blank).



- Select a rectangular workpiece blank: Press the soft key for a rectangular blank form

Working plane in graphic: XY

Z

- Enter the spindle axis, e.g. **Z**

Workpiece blank def.: Minimum

ENT

- Enter in sequence the X, Y and Z coordinates of the MIN point and confirm each of your entries with the **ENT** key

Workpiece blank def.: Maximum

ENT

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

Example

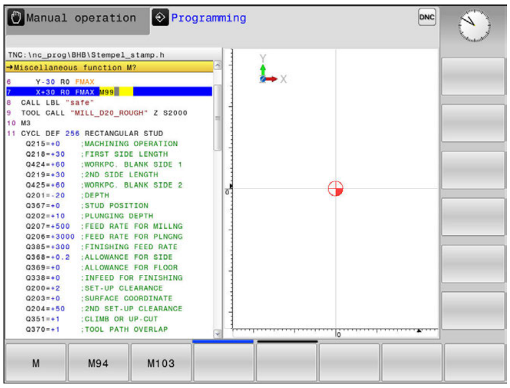
0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Spindle axis, MIN point coordinates
2 BLK FORM 0.2 X+100 Y+100 Z+0	MAX point coordinates
3 END PGM NEW MM	Program end, name, unit of measure

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

> If you do not wish to define a blank form, cancel the dialog at **Working plane in graphic: XY** using the **DEL** key.

Programming tool movements in Klartext

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



Example of a positioning block

COORDINATES ?

X

- **10** (enter the target coordinate for the X axis)

ENT

- Go to the next question with **ENT**.

TOOL RADIUS COMP: R+/R-/NO COMP:?

ENT

- Enter **No radius compensation** and go to the next question with **ENT**

Feed rate F=? / F MAX = ENT

- **100** (enter a feed rate of 100 mm/min for this path contour)

ENT

- Go to the next question with **ENT**.

MISCELLANEOUS FUNCTION M ?

- **3** (enter the miscellaneous function **M3 Spindle on**)

END

- With the **END** key, the control ends this dialog.

Example

```
3 X+10  R0 F100 M3
```

Possible feed rate input

Soft key	Functions for setting the feed rate
F MAX	Rapid traverse, blockwise
F AUTO	Traverse feed rate automatically calculated in **TOOL CALL**
F	Move at the programmed feed rate (unit of measure is mm/min or 1/10 inch/min). With rotary axes, the control interprets the feed rate in degrees/min, regardless of whether the program is written in mm or inches
FU	Define the feed per revolution (units in mm/1 or inch/1). Caution: In inch-programs, FU cannot be combined with M136
FZ	Define the tooth feed (units in mm/tooth or inch/tooth). The number of teeth must be defined in the tool table in the **CUT** column.

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

Actual position capture

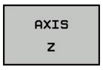
The control enables you to transfer the current tool position into the program, for example during

- Positioning-block programming
- Cycle programming

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 control displays the axes whose positions can be transferred.



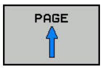
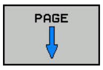
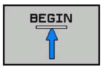
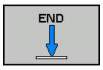
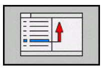
- ▶ Select the axis
- > The control writes the current position of the selected axis into the active input box.

> In the working plane the control always captures the coordinates of the tool center, even though tool radius compensation is active.
>
> The control takes the active tool length compensation into account and always captures the coordinate of the tool tip in the tool axis.
>
> The control keeps the soft-key row for axis selection active until the **actual position capture** key is pressed again. This behavior remains in effect even if you save the current block or open a new block with a axis key. If you have to choose an input alternative via soft key (e.g. for radius compensation), then the control closes the soft-key row for axis selection.

Editing an NC program

> The active NC program cannot be edited while it is being run.

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

Soft key/key	Function
	Go to previous page
	Go to next page
	Go to beginning of program
	Go to end of program
	Change the position of the current block on the screen. Press this soft key to display additional NC blocks that are programmed before the current block No function if the NC program is fully visible on the screen
	Change the position of the current block on the screen. Press this soft key to display additional NC blocks that are programmed after the current block No function if the NC program is fully visible on the screen
 	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: Press the **GOTO** key, enter the block number step and jump up or down the number of entered lines by pressing the **N LINES** soft key

Soft key/key	Function
CE	■ Set the selected word to zero ■ Erase an incorrect number ■ Delete the (clearable) error message
NO ENT	Delete the selected word
DEL	■ Delete the selected block ■ Erase cycles and program sections
INSERT LAST NC BLOCK	Insert the block that you last edited or deleted

Inserting blocks at any desired location

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

Saving changes

The control normally saves changes automatically if you switch the operating mode or if you select the file manager. If you deliberately want to save changes to the program, proceed as follows:

▶ Select the soft-key row with the saving functions

STORE

▶ Press the **STORE** soft key

> The control saves all changes made since the last time you saved the program.

Saving a program to a new file

You can save the contents of the currently active program under a different program name. Proceed as follows:

▶ Select the soft-key row with the saving functions

SAVE AS

▶ Press the **SAVE AS** soft key

> The control opens a window in which you can enter the directory and the new file name.

▶ Select the target directory if required with the **SWITCH** soft key

▶ Enter the file name

▶ Confirm with the **OK** soft key or the **ENT** key, or press the **CANCEL** soft key to abort

The file saved with **SAVE AS** can also be found in the file management by pressing the **LAST FILES** soft key.

Undoing changes

You can undo all changes made since the last time you saved the program. Proceed as follows:

- Select the soft-key row with the saving functions

CANCEL CHANGE

- Press the **CANCEL CHANGE** soft key
- > The control opens a window in which you can confirm or cancel this action.
- Confirm with the **YES** soft key or cancel with the **ENT** key, or press the **NO** soft key to abort

Editing and inserting words

- Select a word in a block and overwrite it with the new one. The 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

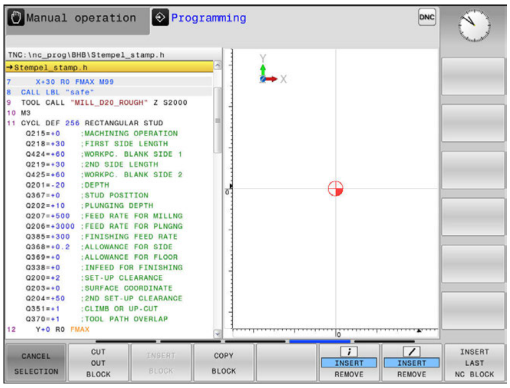
- Select a word in a block: Press the arrow key repeatedly until the desired word is highlighted
- Select a block with the arrow keys
  - Arrow down: search forwards
  - Arrow up: search backwards

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

> If you start a search in a very long NC program, the control shows a progress indicator. You can cancel the search at any time, if necessary.

Marking, copying, cutting and inserting program sections

The control provides the following functions for copying program sections within an NC program or into another NC program:



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

To copy a program section, proceed as follows:

- ▶ Select the soft key row containing the marking functions
- ▶ Select the first block of the section you wish to copy
- ▶ Mark the first block: Press the **SELECT BLOCK** soft key.
- > The control highlights the block in color and displays the **CANCEL SELECTION** soft key.
- ▶ Move the highlight to the last block of the program section you wish to copy or cut.
- > The control shows the marked blocks in a different color. You can end the marking function at any time by pressing the **CANCEL SELECTION** soft key.
- ▶ Copy the selected program section: Press the **COPY BLOCK** soft key. Cut the selected program section: Press the **CUT OUT BLOCK** soft key.
- > The control stores the selected block.

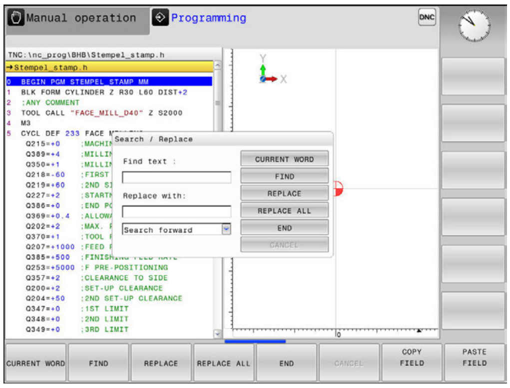
> **i** If you want to transfer a program section to another NC program, you now need to select the desired NC program in the file manager.

- ▶ Using the arrow keys, select the block after which you wish to insert the copied (cut) program section
- ▶ Insert the saved program section: Press the **INSERT BLOCK** soft key
- ▶ To end the marking function, press the **CANCEL SELECTION** soft key

The control's search function

The search function of the control enables you to search for any text within a program and replace it by a new text, if required.

Finding any text



FIND

- ▶ Select the search function
- > The control superimposes the search window and displays the available search functions in the soft-key row.
- ▶ Enter the text to be searched for, e.g.: **TOOL**
- ▶ Select forwards search or backwards search

FIND

- ▶ Start the search process
- > The control moves to the next block containing the text you are searching for.

FIND

- ▶ Repeat the search process
- > The control moves to the next block containing the text you are searching for.

END

- ▶ Terminate the search function: Press the END soft key

Finding/Replacing any text

NOTICE

Caution: Data may be lost!

The **REPLACE** and **REPLACE ALL** functions overwrite all found syntax elements without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a consequence, NC programs may be irreversibly damaged.

- Back up the NC programs, if required, before you start the replacement
- Be careful when using **REPLACE** and **REPLACE ALL**

The **FIND** and **REPLACE** functions cannot be used in the active NC program while the program is being run. The functions are also not available if write protection is active.

- Select the block containing the word you wish to find

FIND

- Select the search function
- > The control superimposes the search window and displays the available search functions in the soft-key row.
- Press the **CURRENT WORD** soft key
- > The control loads the first word of the current block. If required, press the soft key again to load the desired word.

FIND

- Start the search process
- > The control moves to the next occurrence of the text you are searching for.

REPLACE

- 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 ALL** soft key. To skip the text and move to its next occurrence press the **FIND** soft key

END

- Terminate the search function: Press the END soft key

3.3 File management: Basics

Files

Files in the control	Type
Programs	
in HEIDENHAIN format	.H
Tables for	
Tools	.T
Tool changers	.TCH
Datums	.D
Points	.PNT
Presets	.PR
Touch probes	.TP
Backup files	.BAK
Dependent data (e.g. structure items)	.DEP
Freely definable tables	.TAB
Text as	
ASCII files	.A
Log files	.TXT
Help files	.CHM

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

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

With the control you can manage and save files up to a total size of **2 GB.**

> Depending on the setting, the control generates backup files with the extension *.bak after editing and saving of NC programs. This reduces the available memory space.

File names

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

File name	File type
PROG20	.H

File names, drive names and directory names on the control must comply with the following standard: The Open Group Base Specifications Issue 6 IEEE Std 1003.1, 2004 Edition (POSIX Standard).

The following characters are permitted:

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _ -

The following characters have special meanings:

Character	Meaning
.	The last period (dot) in a file name is the extension separator
\ and **/**	Directory separators
:	Separates the drive name from the directory

Do not use any other characters. This helps to prevent file transfer problems, etc. Table names must start with a letter.

> The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.
>
> **Further information:** "Paths", page 128

Displaying externally generated files on the control

The control features several additional tools which you can use to display the files shown in the table below. Some of the files can also be edited.

File types	Type
PDF files Excel tables Internet files	pdf xls csv html
Text files	txt ini
Graphics files	bmp gif jpg png

Further information: "Additional tools for management of external file types", page 141

Data backup

HEIDENHAIN recommends backing up new programs and files created on the control to a PC at regular intervals.

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

You can also backup files directly from the control. **Further information:** "Backup and restore", page 102

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.

> Take the time occasionally to delete any unneeded files so that the control always has enough hard-disk space for system files (such as the tool table).

3.4 Working with the file manager

Directories

To ensure that you can easily find your programs and files, we recommend that you organize your internal memory 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.

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 ****.

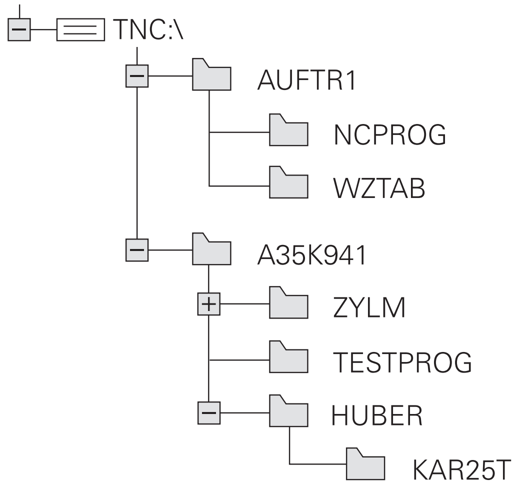
> The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

Example

The directory AUFTR1 was created on the **TNC** drive. Then, in the AUFTR1 directory, 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.



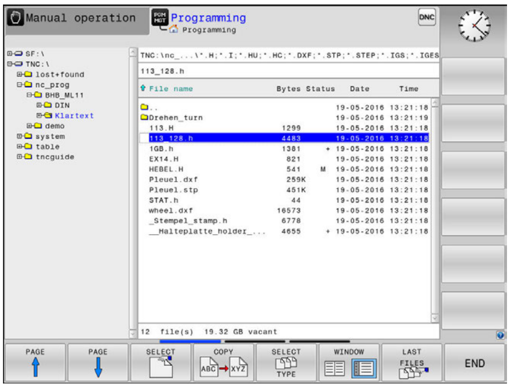
Overview: Functions of the file manager

Soft key	Function	Page
COPY ABC → XYZ	Copy a single file	133
SELECT TYPE	Display a specific file type	131
NEW FILE	Create new file	133
LAST FILES	Display the last 10 files that were selected	136
DELETE	Delete a file	136
TAG	Tag a file	138
RENAME ABC = XYZ	Rename file	139
PROTECT	Protect a file against editing and erasure	140
UNPROTECT	Cancel file protection	140
ADAPT NC PGM / TABLE	Import tool table of an iTNC 530	197
	Customize table view	355
NET	Manage network drives	150
SELECT EDITOR	Select the editor	140
SORT	Sort files by properties	139
COPY DIR	Copy a directory	136
DELETE ALL	Delete directory with all its subdirectories	
UPDATE TREE	Refresh directory	
RENAME ABC = XYZ	Rename a directory	
NEW DIRECTORY	Create a new directory	

Calling the file manager

PGM MGT

- Press the **PGM MGT** key
- > The control displays the file management window (see figure for default setting. If the control 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. A drive is the internal memory of the control. Other drives are the interfaces (RS232, Ethernet) to which you can connect a PC for example. A directory is always identified by a folder symbol to the left and the directory name to the right. Subdirectories are shown to the right of and below their parent directories. If there are subdirectories, you can show or hide them using the **-/+** key.

If the directory tree is longer than the screen, navigate using the scroll bar or a connected mouse.

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	File name and file type
Bytes	File size in bytes
Status	File properties:
E	Program is selected in the **Programming** 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
+	Program has non-displayed dependent files with the extension DEP, e.g. with use of the tool usage test
	File is protected against erasing and editing
	File is protected against erasing and editing, because it is being run
Date	Date that the file was last edited
Time	Time that the file was last edited

> To display the dependent files, set the machine parameter **dependentFiles** (no. 122101) to **MANUAL**.

Selecting drives, directories and files

PGM MGT

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

Navigate with a connected mouse or use the arrow keys or the soft keys to move the cursor to the desired position on the screen:

- Moves the cursor from the left to the right window, and vice versa
- Moves the cursor up and down within a window

PAGE / PAGE

- Moves the cursor 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

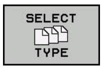
ENT

- Press the **ENT** key

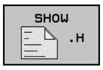
Step 2: Select a directory

- 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

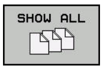
Step 3: Select a file



- Press the **SELECT TYPE** soft key



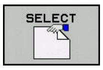
- Press the soft key for the desired file type, or



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

- Use wildcards, e.g. **4*.h**: Show all files of type .h starting with a 4

- Move the highlight to the desired file in the right window



- Press the **SELECT** soft key, or

- Press the **ENT** key
- > The control opens the selected file in the operating mode from which you called the file manager.

> **i** If you enter the first letter of the file you are looking for in file management, the cursor automatically jumps to the first program with the same letter.

Creating a new directory

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

- Press the **NEW DIRECTORY** soft key
- Enter a directory name

- Press the **ENT** key

- Press the **OK** soft key to confirm or

- Press the **CANCEL** soft key to abort

Creating new file

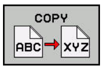
- Select the directory in the left window in which you wish to create the new file
- Position the cursor in the right window

- Press the **NEW FILE** soft key
- Enter the file name with extension

- Press the **ENT** key

Copying a single file

- Move the cursor to the file you wish to copy



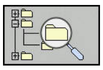
- Press the **COPY** soft key to select the copying function
- > The control opens a pop-up window.

Copying files into the current directory

- Enter the name of the destination file.

- Press the **ENT** key or the **OK** soft key
- > The control copies the file to the active directory. The original file is retained.

Copying files into another directory



- Press the **Target Directory** soft key to select the target directory from a pop-up window

- Press the **ENT** key or the **OK** soft key
- > The control copies the file under the same name to the selected directory. The original file is retained.

> When you start the copying process with the **ENT** key or the **OK** soft key, the control displays a pop-up window with a progress indicator.

Copying files into another directory

- Select a screen layout with two equally sized windows

In the right window

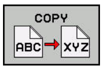
- Press the **SHOW TREE** soft key
- Move the cursor to the directory into which you wish to copy the files,

In the left window

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

- Press the Tag soft key: Call the file tagging functions

- Press the Tag soft key: Position the cursor on the file you wish to copy and tag. You can tag several files in this way, if desired



- Press the Copy soft key: Copy the tagged files into the target directory

Further information: "Tagging files", page 138

If you have tagged files in both the left and right windows, the control copies from the directory in which the cursor is located.

Overwriting files

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

- Overwrite all files (**Existing files** field selected): Press the **OK** soft key, or
- To leave the files as they are, press the **CANCEL** soft key

If you want to overwrite a protected file, select the **Protected files** field or cancel the process.

Copying a table

Importing lines to a table

If you are copying a table into an existing table, you can overwrite individual lines with the **REPLACE FIELDS** soft key. Prerequisites:

- The target table must exist
- The file to be copied must only contain the lines you want to replace
- Both tables must have the same file extension

NOTICE

Caution: Data may be lost!

If you use the **REPLACE FIELDS** function, all lines of the target file that are contained in the copied table will be overwritten without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a consequence, tables may be irreversibly damaged.

- ▶ Back up the tables, if required, before you start the replacement
- ▶ Be careful when using **REPLACE FIELDS**

Example

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

- ▶ Copy this table from the external data medium to any directory
- ▶ Copy the externally created table to the existing TOOL.T table using the control's file management.
- > The control asks you whether you want to overwrite the existing TOOL.T tool table.
- ▶ If you press the **REPLACE FIELDS** soft key, the control will completely overwrite the current TOOL.T tool table. After this copying process the new TOOL.T table consists of 10 lines.
- ▶ Or press the **REPLACE FIELDS** soft key for the control to overwrite the 10 lines in the TOOL.T file. The data of the other lines is not changed.

Extracting lines from a table

You can select one or more lines in a table and save them in a separate table.

- ▶ Open the table from which you want to copy lines
- ▶ Use the arrow keys to select the first line to be copied
- ▶ Press the **MORE FUNCTIONS** soft key
- ▶ Press the **TAG** soft key
- ▶ Select additional lines, if required
- ▶ Press the **SAVE AS** soft key
- ▶ Enter a name for the table in which the selected lines are to be saved

Copying a directory

- Move the highlight in the right window onto the directory you want to copy
- Press the **COPY** soft key
- > The control opens the window for selecting the target directory.
- Select the target directory and confirm with the **ENT** key or the **OK** soft key
- > The control copies the selected directory and all its subdirectories to the selected target directory.

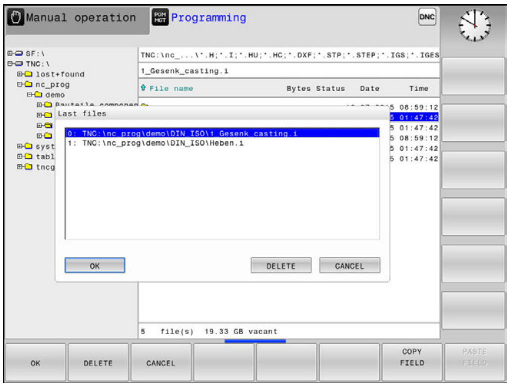
Choosing one of the last files selected

PGM MGT

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

LAST FILES

- To display the last ten files selected: press the **LAST FILES** soft key



Press the arrow keys to move the cursor to the file you wish to select:

- Moves the cursor up and down within a window

OK

- To select the file, press the **OK** soft key, or

ENT

- Press the **ENT** key

> The **COPY FIELD** soft key allows you to copy the path of a marked file. You can reuse the copied path later, e.g. when calling a program with the **PGM CALL** key.

Deleting a file

NOTICE

Caution: Data may be lost!

The **DELETE** function permanently deletes the file. The file is not automatically backed up by the control, e.g. to a recycle bin, before being deleted. Files are irreversibly deleted by this function.

- Regularly back up important data to external drives

- Move the cursor to the file you want to delete

DELETE

- To select the erasing function, press the **DELETE** soft key
- > The control asks whether you want to delete the file.
- To confirm the deletion, press the **OK** soft key; or
- To cancel deletion, press the **CANCEL** soft key

Deleting a directory

NOTICE

Caution: Data may be lost!

The **DELETE ALL** function permanently deletes all files of the directory. The files are not automatically backed up by the control, e.g. to a recycle bin, before being deleted. Files are irreversibly deleted by this function.

- Regularly back up important data to external drives

- Move the cursor to the directory you want to delete

- To select the erasing function, press the **DELETE** soft key
- \> The control asks you whether you really want to delete the directory and all its subdirectories and files.
- To confirm the deletion, press the **OK** soft key; or
- To cancel deletion, press the **CANCEL** soft key

Tagging files

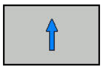
Soft key	Tagging function
	Tag a single file
	Tag all files in the directory
	Untag a single file
	Untag all files

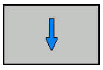
Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To tag several files, proceed as follows:

- Move the cursor to the first file

- To display the tagging functions, press the **TAG** soft key

- To tag the file, press the **TAG FILE** soft key

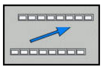




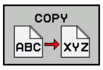
- Move the cursor to other files

- To select the next file, press the **TAG FILE** soft key. Repeat this process for all files you want to tag.

To copy tagged files:

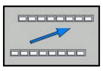


- Leave the active soft-key row



- Press the **COPY** soft key

To delete tagged files:

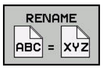


- Leave the active soft-key row

- Press the **DELETE** soft key

Renaming a file

- Move the cursor to the file you wish to rename



- To select the function for renaming, press the **RENAME** soft key
- Enter the new file name; the file type cannot be changed
- To rename: Press the **OK** soft key or the **ENT** key

Sorting files

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

- Press the **SORT** soft key
- Select the soft key with the corresponding display criterion
  - **SORT BY NAME**
  - **SORT BY SIZE**
  - **SORT BY DATE**
  - **SORT BY TYPE**
  - **SORT BY STATUS**
  - **UNSORTED**

Additional functions

Protecting a file / Canceling file protection

- Move the cursor to the file you want to protect

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

- Enable file protection: Press the **PROTECT** soft key. The file is tagged with the "protected" symbol

- To cancel file protection, press the **UNPROTECT** soft key

Selecting the editor

- Move the cursor in the right-hand window onto the file you want to open

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

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

Connecting and removing USB storage devices

The control automatically detects connected USB devices with a supported file system.

- To remove a USB device, proceed as follows:

- Move the cursor to the left-hand window
- Press the **MORE FUNCTIONS** soft key

- Remove the USB device

Further information: "USB devices on the control", page 151

Additional tools for management of external file types

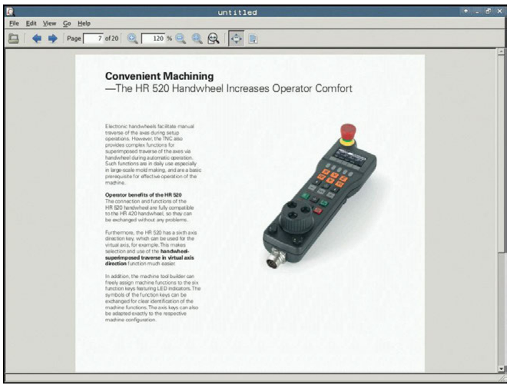
The additional tools enable you to display or edit various externally created file types on the control.

File types	Description
PDF files (pdf)	page 142
Excel spreadsheets (xls, csv)	page 143
Internet files (htm, html)	page 144
ZIP archives (zip)	page 146
Text files (ASCII files, e.g. txt, ini)	page 147
Video files (ogg, oga, ogv, ogx)	page 148
Graphics files (bmp, jpg, gif, png)	page 148

Files with the extensions pdf, xls, zip, bmp, gif, jpg and png must be transmitted in binary format from the PC to the control. Adjust the setting in the TNCremo data transfer software, if required (menu item >**Extras > Configuration > Mode**).

Displaying PDF files

To open PDF files directly on the control, proceed as follows:



PGM MGT

- ▶ To call the file manager, press the **PGM MGT** key.
- ▶ Select the directory in which the PDF file is saved
- ▶ Move the cursor to the PDF file

ENT

- ▶ Press the **ENT** key
- > The control opens the PDF file in its own application using the **PDF viewer** additional tool.

> With the key combination ALT+TAB you can always return to the control's user interface while leaving the PDF file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the user interface of the control.

> If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use the **PDF viewer** is provided under **Help**.

Proceed as follows to exit the **PDF viewer**:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select **Close**
- > The control returns to the file management.

If you are not using a mouse, proceed as follows to close the **PDF viewer**:

- ▶ Press the key for switching the soft keys
- > The **PDF viewer** opens the **File** pull-down menu.
- ▶ Move the cursor to the **Close** menu item.

ENT

- ▶ Press the **ENT** key
- > The control returns to the file management.

Displaying and editing Excel files

Proceed as follows to open and edit Excel files with the extension **xls**, **xlsx** or **csv** directly on the control:

PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Select the directory in which the Excel file is saved
- Move the cursor to the Excel file

ENT

- Press the **ENT** key
- \> The control opens the Excel file in its own application using the **Gnumeric** additional tool.

> With the key combination ALT+TAB you can always return to the control's user interface while leaving the Excel file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the user interface of the control.

> If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use the **Gnumeric** function is provided under **Help**.

Proceed as follows to exit **Gnumeric**:

- Use the mouse to select the **File** menu item
- Select **Close**
- \> The control returns to the file management.

If you are not using a mouse, proceed as follows to close the additional **Gnumeric** tool:

▷

- Press the key for switching the soft keys
- \> The **Gnumeric** additional tool opens the **File** pull-down menu.

↓

- Move the cursor to the **Close** menu item

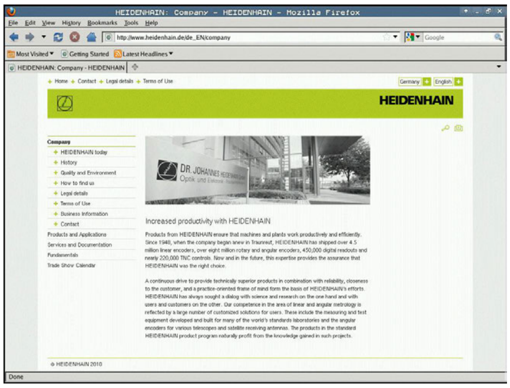
ENT

- Press the **ENT** key
- \> The control returns to the file management.

Displaying Internet files

Configure and use the sandbox on your control. For safety and security reasons, always open the browser in the sandbox.

Proceed as follows to open Internet files with the extension **htm** or **html** directly on the control:



PGM MGT

- ▶ To call the file manager, press the **PGM MGT** key.
- ▶ Select the directory in which the Internet file is saved
- ▶ Move the cursor to the Internet file

ENT

- ▶ Press the **ENT** key
- > The control opens the Internet file in its own application using the **Web Browser** additional tool.

With the key combination ALT+TAB you can always return to the control's user interface while leaving the PDF file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the user interface of the control.

If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use **Web Browser** is available in **Help**.

Proceed as follows to exit the **Web Browser**:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select **Quit**
- > The control returns to the file management.

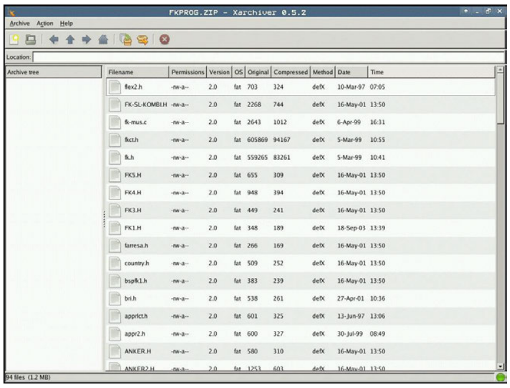
If you are not using a mouse, proceed as follows to close the **Web Browser**:

- ▶ Press the key for switching the soft keys: The **Web Browser** opens the **File** pull-down menu
- ▶ Move the cursor to the **Quit** menu item
- ▶ Press the **ENT** key
- > The control returns to the file management.

> Do not change the Web Browser version.
> Otherwise, the security settings of SELinux will block the execution of Web Browser.

Working with ZIP archives

Proceed as follows to open ZIP archives with the extension **zip** directly on the control:



PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Select the directory in which the archive file is saved
- Move the cursor to the archive file

ENT

- Press the **ENT** key
- > The control opens the archive file in its own application using the **Xarchiver** additional tool.

> With the key combination ALT+TAB you can always return to the control's user interface while leaving the archive file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the user interface of the control.

> If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use the **Xarchiver** function is provided under **Help**.

Proceed as follows to exit **Xarchiver**:

- Use the mouse to select the **ARCHIVE** menu item
- Select **Exit**
- > The control returns to the file management.

If you are not using a mouse, proceed as follows to close the **Xarchiver**:

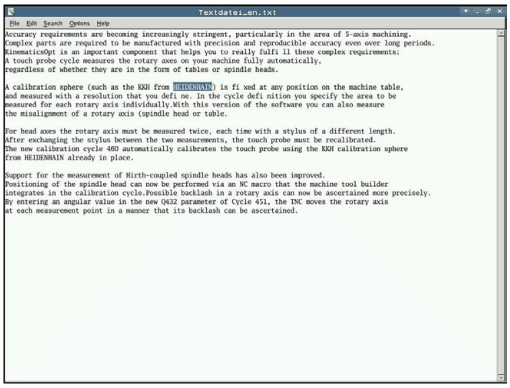
- Press the key for switching the soft keys
- > **Xarchiver** opens the **ARCHIVE** pull-down menu.
- Move the cursor to the **Exit** menu item

ENT

- Press the **ENT** key
- > The control returns to the file management.

Displaying and editing text files

Use the internal text editor to open and edit text files (ASCII files, e.g. with the extension **txt**). Proceed as follows:



PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Select the drive and the directory in which the text file is saved
- Move the cursor to the text file

ENT

- Press the **ENT** key
- > The control opens the text file with the internal text editor.

Alternatively, you can also open the ASCII files using the **Leafpad** additional tool. The shortcuts you are familiar with from Windows, which you can use to edit texts quickly (CTRL+C, CTRL+V,...), are available within **Leafpad**.

With the key combination ALT+TAB you can always return to the control's user interface while leaving the text file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the user interface of the control.

Proceed as follows to open **Leafpad**:

- Use the mouse to select the **Menu** HEIDENHAIN icon from the task bar
- Select the **Tools** and **Leafpad** menu items in the pull-down menu

Proceed as follows to exit **Leafpad**:

- Use the mouse to select the **File** menu item
- Select **Exit**
- > The control returns to the file management.

Displaying video files

This feature must be enabled and adapted by the machine tool builder.

Proceed as follows to open video files with the extension **ogg**, **oga**, **ogv** or **ogx** directly on the control:

PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Select the directory in which the video file is saved
- Move the cursor to the video file

ENT

- Press the **ENT** key
- > The control opens the video file in its own application.

Displaying graphic files

Proceed as follows to open graphics files with the extension **bmp**, **gif**, **jpg** or **png** directly on the control:

PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Select the directory in which the graphics file is saved
- Move the cursor to the graphics file

ENT

- Press the **ENT** key
- > The control opens the graphics file in its own application using the **ristretto** additional tool.

With the key combination ALT+TAB you can always return to the control's user interface while leaving the graphics file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the user interface of the control.

More information on how to use the **ristretto** function is provided under **Help**.

Proceed as follows to exit **ristretto**:

- Use the mouse to select the **File** menu item
- Select **Exit**
- > The control returns to the file management.

If you are not using a mouse, proceed as follows to close the additional **ristretto** tool:

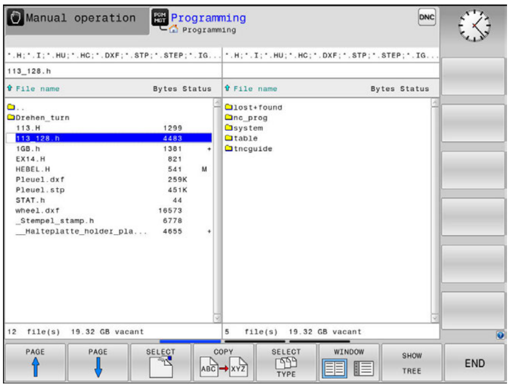
- Press the key for switching the soft keys
- > **ristretto** opens the **File** pull-down menu.
- Move the cursor to the **Exit** menu item

ENT

- Press the **ENT** key
- > The control returns to the file management.

Data transfer to or from an external data carrier

Before you can transfer data to an external data medium, you must set up the data interface.
Further information: "Setting up data interfaces", page 479



PGM MGT

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

WINDOW

- Select the screen layout for data transfer: press the **WINDOW** soft key.

Use the arrow keys to move the cursor to the file you wish to transfer:

- Moves the cursor up and down within a window

- Moves the cursor from the right to the left window, and vice versa

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

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

SHOW TREE

- Select another drive or directory: Press the **SHOW TREE** soft key
- Use the arrow keys to select the desired directory

SHOW FILES

- Select the desired file: Press the **SHOW FILES** soft key

COPY ABC → XYZ

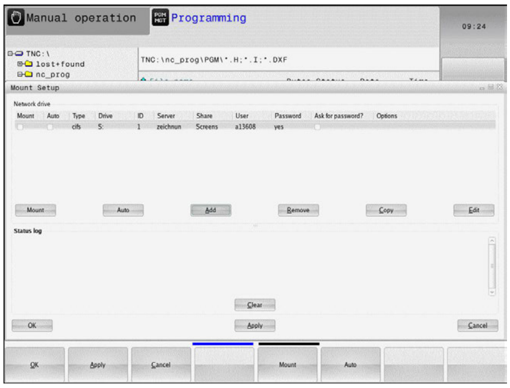
- Use the arrow keys to select the file
- Transfer a single file: Press the **COPY** soft key

- Confirm with the **OK** soft key or with the **ENT** key
- > A status window appears on the control, informing about the copying progress, or

WINDOW

- Stop transfer: Press the **WINDOW** soft key
- > The control displays the standard file manager window again.

The control in a network



> Protect your data and your control by running your machines in a secure network.

> Use the Ethernet card to connect the control to the network.
> **Further information:** "Ethernet interface ", page 485
> The control logs any error messages that occur during network operation.

If the control is connected to a network, the left directory window displays additional drives. All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been granted the corresponding rights.

Connecting and disconnecting a network drive

PGM MGT

- To call the file manager, press the **PGM MGT** key

NET

- Select network settings: Press the **NET** soft key (soft-key row 2)
- To manage the network drives: Press the **DEFINE NETWORK CONNECTN.** soft key.
- > In a window the control shows the network drives available for access.
- With the soft keys described below you can define the connection for each drive.

Soft key	Function
Connect	Establish the network connection. If the connection is active, the control marks the **Mount** column.
Separate	End network connection
Auto	Automatically establish network connection whenever the control is switched on. The control marks the **Auto** column if the connection is established automatically
Add	Set up new network connection
Remove	Delete existing network connection
Copy	Copy network connection
Edit	Edit network connection
Clear	Delete the status window

USB devices on the control

> Use the USB port only for file transfer and backup. Before editing or running an NC program, save it to the hard disk of the control. This helps to avoid duplicate data maintenance and prevents potential problems resulting from data transfer during program run.

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

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

> If an error message is displayed when connecting a USB data medium, check the setting in the SELinux security software.
>
> **Further information:** "SELinux security software", page 99
>
> If the control displays the **USB: TNC does not support device** error message when using a USB hub, ignore and acknowledge the message with the **CE** key.
>
> If the control repeatedly fails to correctly detect a USB device with the FAT/VFAT file system, connect another device to check the port. If this resolves the problem, use the properly working device.

Working with USB devices

> Refer to your machine manual.
>
> Your machine tool builder can assign permanent names for USB devices.

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

If a larger file is transferred to a USB device in the file management, the control displays a dialog **Write access on USB device** until file transfer is completed. The dialog is closed with the **HIDE** soft key and file transfer is continued in the background. The control displays a warning until file transfer is completed.

Removing USB devices

- To remove a USB device, proceed as follows:

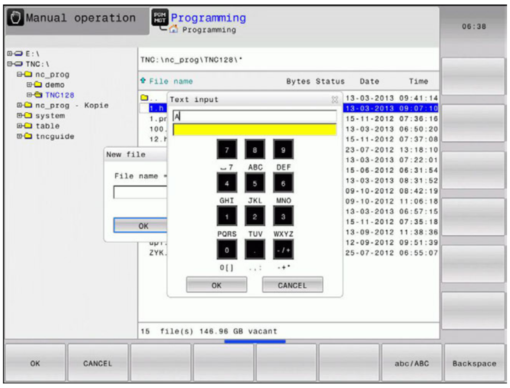
- Move the cursor to the left-hand window
- Press the **MORE FUNCTIONS** soft key
- Remove the USB device

4

Programming Aids

4.1 Screen keypad

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



Entering text with the screen keyboard

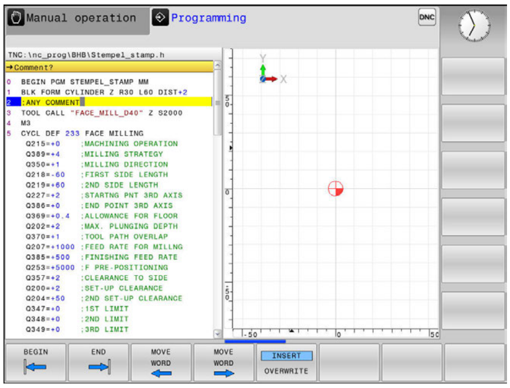
- Press the **GOTO** key if you want to enter letters, e.g. a program name or directory name, using the screen keypad.
- > The control opens a window in which the numeric entry field of the control is displayed with the corresponding letters assigned.
- You can move the cursor to the desired character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

Use the **abc/ABC** soft key to select upper or lower case. If your machine manufacturer has defined additional special characters, you can call them with the **SPECIAL CHARACTERS** soft key and insert them. Use the **BACKSPACE** soft key to delete individual characters.

4.2 Adding comments

Application

You can add comments to an NC program to explain program steps or make general notes.



> The control shows long comments in different ways, depending on the machine parameter **lineBreak** (no. 105404). It either wraps the comment lines or displays the **>>** symbol to indicate additional content.
>
> The last character in a comment block must not have any tilde(~).

You can add comments in different ways.

Add comments

- Select the NC block after which you want to insert the comment

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM-MING AIDS

- Press the **PROGRAMMING AIDS** soft key

INSERT COMMENT

- Press the **INSERT COMMENT** soft key
- Enter text

Entering comments during programming

> To use this function you will need a keyboard connected via USB.

- Enter the data for an NC block
- Press the semicolon key **;** on the alphabetic keyboard
- > The control displays the dialog prompt **Comment?**
- Enter the comment
- Press the **END** key to conclude the NC block

Inserting comments after program entry

> To use this function you will need a keyboard connected via USB.

- Select the NC block to which you want to add the comment
- Select the last word in the NC block with the right arrow key:
- Press the semicolon key **;** on the alphabetic keyboard
- > The control displays the dialog prompt **Comment?**
- Enter the comment
- Press the **END** key to conclude the NC block

Entering a comment in a separate block

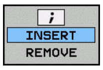
> To use this function you will need a keyboard connected via USB.

- Select the NC block after which you want to insert the comment
- Initiate the programming dialog with the semicolon key **;** on the alphabetic keyboard
- Enter your comment and conclude the NC block by pressing the **END** key

Commenting out an existing NC block

Proceed as follows to change an existing NC block to a comment:

- Select the NC block to be commented out

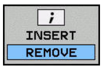


- Press the **INSERT COMMENT** soft key
- > The control inserts a semicolon **;** at the beginning of the block.
- Press the **END** key

Changing a comment for an NC block

Proceed as follows to change a commented-out NC block to an active NC block:

- Select the comment block you want to change

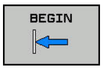
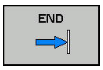
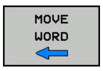
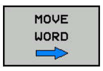
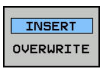


- Press the **REMOVE COMMENT** soft key

Alternative:

- Press the **>** key on the alphabetic keyboard
- > The control removes the semicolon **;** at the beginning of the block.
- Press the **END** key

Functions for editing of the comment

Soft key	Function
	Jump to beginning of comment
	Jump to end of comment
	Jump to the beginning of a word. Use a space to separate words
	Jump to the end of a word. Use a space to separate words
	Switch between paste and overwrite mode

4.3 Freely editing an NC program

Certain syntax elements, such as LN blocks, cannot be entered directly in the NC editor by using the available keys and soft keys.

To prevent the use of an external text editor, the control offers the following possibilities:

- Free syntax input using the control's integrated text editor
- Free syntax input using the **?** key in the NC editor

Free syntax input using the control's integrated text editor

Proceed as follows to add syntax to an existing NC program:

PGM MGT

- ▶ Press the **PGM MGT** key
- > The control opens the file manager.

MORE FUNCTIONS

- ▶ Press the **MORE FUNCTIONS** soft key

SELECT EDITOR

- ▶ Press the **SELECT EDITOR** soft key
- > The control opens a selection window.

OK

- ▶ Select the **TEXT EDITOR** option
- ▶ Confirm your selection with **OK**
- ▶ Add the desired syntax

> The control does not check the syntax in the text editor. Check your entries in the NC editor when you are finished.

Free syntax input using the ? key in the NC editor

> To use this function you will need a keyboard connected via USB.

Proceed as follows to add syntax to an existing, open NC program:

⇧ ?/

- ▶ Enter **?**
- > The control opens a new NC block.

END

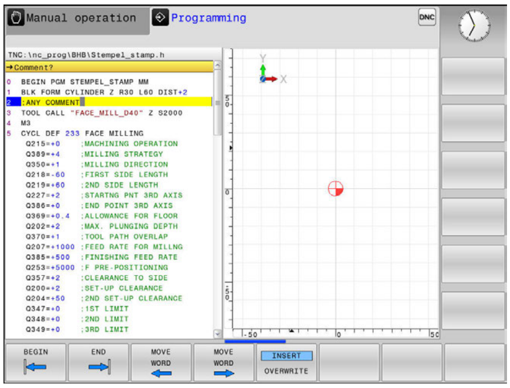
- ▶ Add the desired syntax
- ▶ Confirm your entry with **END**

> After confirmation, the control checks the syntax. Errors will result in **ERROR** blocks.

4.4 Display of NC programs

Syntax highlighting

The control displays syntax elements with various colors according to their meaning. Programs are made more legible and clear with color-highlighting.



Color highlighting of syntax elements

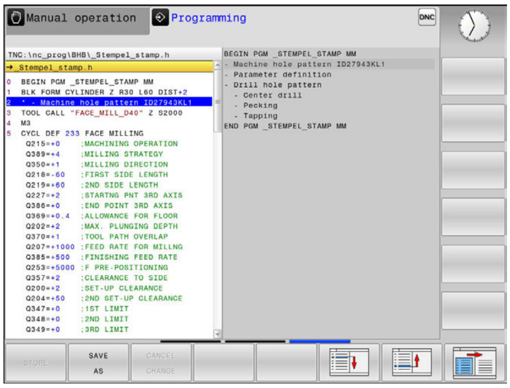
Use	Color
Standard color	Black
Display of comments	Green
Display of numerical values	Blue
Display of the block number	Violet
Display of FMAX	Orange
Display of the feed rate	Brown

Scrollbar

Screen content can be shifted with the mouse using the scroll bar at the right edge of the program window. In addition, the size and position of the scrollbar indicates program length and cursor position.

4.5 Structuring programs

Definition and applications



The control offers you the possibility to comment part programs in structuring blocks. Structuring blocks are texts with up to 252 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 manner.

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.

Structure blocks can also be displayed in a separate window, and be edited or added to, as desired. Use the appropriate screen layout for this.

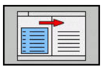
The control manages the inserted structure items in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

The **PROGRAM + SECTS** screen layout can be selected in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Programming**

Displaying the program structure window / Changing the active window

- Display structure window: For this screen layout press the **PROGRAM + STRUCTURE** soft key



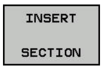
- Change the active window: Press the **CHANGE WINDOW** soft key

Inserting a structure block in the program window

- Select the block after which the structuring block is to be inserted

- Press the **SPEC FCT** key

- Press the **PROGRAMMING AIDS** soft key



- Press the **INSERT SECTION** soft key
- Enter the structuring text

- If necessary, change the structure depth with the soft key

Selecting blocks in the program structure window

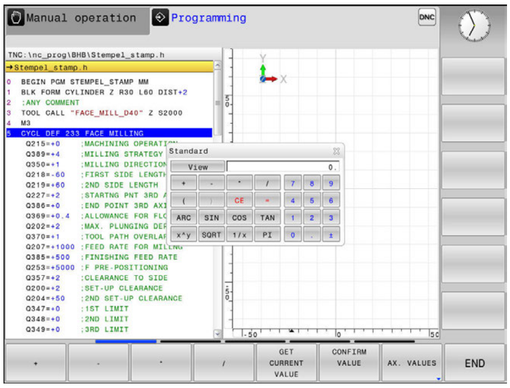
If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

4.6 Calculator

Operation

The control features an integrated calculator with the basic mathematical functions.

- Use the **CALC** key to show and hide the calculator
- Select the arithmetical functions: The calculator is operated with short commands via soft key or through the alphabetic keyboard



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

Calculate function	Shortcut (soft key)
Truncate decimal places	INT
Truncate places before the decimal point	FRAC
Modulus operator	MOD
Select view	View
Delete value	CE
Unit of measure	MM or INCH
Show angle values in radians (standard: angle in degrees)	RAD
Select the display mode of the numerical value	DEC (decimal) or HEX (hexadecimal)

Transferring the calculated value into the program

- Use the arrow keys to select the word into which the calculated value is to be transferred
- Superimpose the on-line calculator by pressing the **CALC** key and perform the desired calculation
- Press the **CONFIRM VALUE** soft key
- > The control transfers the value into the active input field and closes the calculator.

> You can also transfer values from an NC program into the calculator. When you press the **GET CURRENT VALUE** soft key or the **GOTO** key, the control transfers the value from the active input field to the calculator.
>
> The calculator remains in effect even after a change in operating modes. Press the **END** soft key to close the calculator.

Functions in the pocket calculator

Soft key	Function
AX. VALUES	Load the nominal or reference value of the respective axis position into the calculator
GET CURRENT VALUE	Load the numerical value from the active input field into the calculator
CONFIRM VALUE	Load the numerical value from the calculator field into the active input field
COPY FIELD	Copy the numerical value from the calculator
PASTE FIELD	Insert the copied numerical value into the calculator
CUTTING DATA CALCULATOR	Open the cutting data calculator

You can also shift the calculator with the arrow keys on your keyboard. If you have connected a mouse you can also position the calculator with this.

4.7 Cutting data calculator

Application

With the cutting data calculator you can calculate the spindle speed and the feed rate for a machining process. Then you can load the calculated values into an opened feed rate or spindle speed dialog box in the NC program.

To open the cutting data calculator, press the **CUTTING DATA CALCULATOR** soft key. The control shows the soft key if you

- open the on-line calculator (press the **CALC** soft key)
- open the dialog field for spindle speed input in the TOOL CALL block
- open the dialog field for feed rate input in positioning blocks or cycles
- enter a feed rate in manual mode (press the **F** soft key)
- enter a spindle speed in manual mode (press the **S** soft key)

The cutting data calculator is displayed with different input fields depending on whether you calculate a spindle speed or a feed rate:

Window for spindle speed calculation:

Code letter	Meaning
R:	Tool radius (mm)
VC:	Cutting speed (m/min)
S=	Result for spindle speed (rev/min)

Window for feed rate calculation:

Code letter	Meaning
S:	Spindle speed (rpm)
Z:	Number of teeth on the tool (n)
FZ:	Feed per tooth (mm/tooth)
FU:	Feed rate per revolution (mm/1)
F=	Result for feed rate (mm/min)

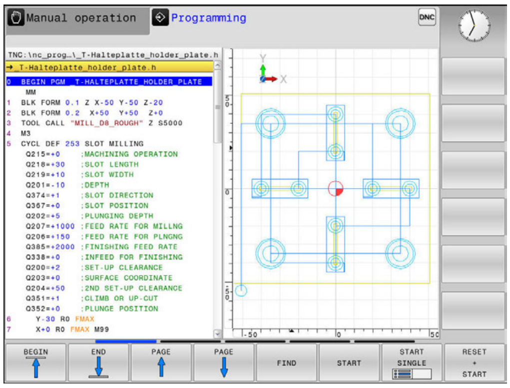
> You can transfer the feed rate from the **TOOL CALL** block into subsequent traversing blocks and cycles by pressing the **F AUTO** soft key. If you have to change the feed rate later, you only need to adjust the feed rate value in the **TOOL CALL** block.

Functions in the cutting data calculator:

Soft key	Function
S RPM	Load the spindle speed from the cutting data calculator form into an open dialog field.
F MM/MIN	Load the feed rate from the cutting data calculator form into an open dialog field.
VC M/MIN	Load the cutting speed from the cutting data calculator form into an open dialog field.
FZ MM/TOOTH	Load the feed per tooth from the cutting data calculator form into an open dialog field.
FU MM/REV	Load the feed per revolution from the cutting data calculator form into an open dialog field.
ACCEPT TOOL RADIUS	Load the tool radius into the cutting data calculator form
CONFIRM RPM	Load the spindle speed from the open dialog field into the cutting data calculator form
ACCEPT FEED RATE	Load the feed rate from the open dialog field into the cutting data calculator form
ACCEPT FEED RATE	Load the feed per revolution from the open dialog field into the cutting data calculator form
ACCEPT FEED RATE	Load the feed per tooth from the open dialog field into the cutting data calculator form
GET CURRENT VALUE	Load the value from an open dialog field into the cutting data calculator form
POCKET CALCULATOR	Switch to the pocket calculator
↓	Move the cutting data calculator in the direction of the arrow
INCH	Use inch values in the cutting data calculator
END	Close the cutting data calculator

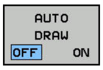
4.8 Programming graphics

Activating and deactivating programming graphics



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

- ▶ Press the **Screen layout** key
- ▶ Press the **PROGRAM + GRAPHICS** soft key
- > The control shows the NC program to the left and graphics to the right.



- ▶ Set the **AUTO DRAW** soft key to **ON**
- > While you are entering the program lines, the control generates each programmed movement in the graphics window in the right screen half.

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

> If **AUTO DRAW** is set to **ON**, the control ignores the following program content when creating 2-D pencil-trace graphics:
> - Program section repetitions
> - Jump commands
> - M functions, such as M2 or M30
> - Cycle calls
> - Warnings due to locked tools
>
> Therefore, only use automatic drawing during contour programming.

The control resets the tool data if you reopen a program or press the **RESET + START** soft key.

The control uses various colors in the programming graphics:

- **blue:** uniquely specified contour element
- **violet:** not yet uniquely specified contour element
- **light blue:** holes and threads
- **ocher:** tool midpoint path
- **red:** rapid traverse

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

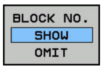
- Reset previously active tool data and generate graphics: Press the **RESET + START** soft key

Additional functions:

Soft key	Function
RESET + START	Reset previously active tool data. Generate programming graphics
START SINGLE	Generate programming graphic blockwise
START	Generate a complete graphic or complete it after **RESET + START**
STOP	Stop the programming graphics. This soft key only appears while the control is generating the programming graphics
VIEWS	Selecting views ■ Plan view ■ Front view ■ Page view
TOOL PATH: SHOW HIDE	Display or hide tool paths
FMAX PATHS DISPLAY HIDE	Display or hide tool paths in rapid traverse

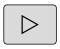
Block number display ON/OFF

- Shift the soft-key row



- To show block numbers: Set the **BLOCK NO. SHOW OMIT** soft key to **SHOW**
- To hide block numbers: Set the **BLOCK NO. SHOW OMIT** soft key to **OMIT**

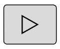
Erasing the graphic



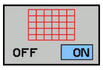
- Shift the soft-key row

- Erase the graphics: Press the **CLEAR GRAPHICS** soft key

Showing grid lines



- Shift the soft-key row

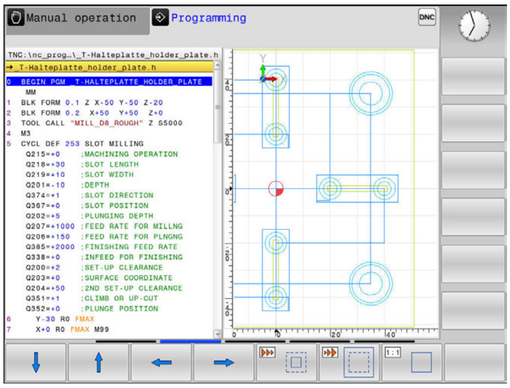


- Show grid lines: Press the **Show grid lines** soft key

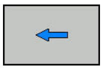
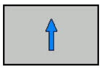
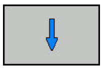
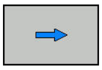
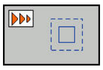
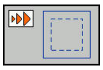
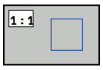
Magnification or reduction of details

You can select the graphics display

- ▶ Shift the soft-key row



The following functions are available:

Soft key	Function
   	Shift section
	Reduce section
	Enlarge section
	Reset section

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

You can also use the mouse to change the graphic display. The following functions are available:

- To shift the model shown, hold the center mouse button or mouse wheel down and move the mouse. If you simultaneously press the shift key, you can only shift the model horizontally or vertically.
- To zoom in on a certain area, mark a zoom area by holding the left mouse button down. After you release the left mouse button, the control zooms in on the defined area.
- To rapidly magnify or reduce any area, rotate the mouse wheel backwards or forwards.

4.9 Error messages

Display of errors

The control displays error messages in the following cases, for example:

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

When an error occurs, the control displays it in red type in the header.

> The control uses different colors for different error classes:
> - red for errors
> - yellow for warnings
> - green for notes
> - blue for information

Long and multi-line error messages are displayed in abbreviated form. Complete information on all pending errors is shown in the error window.

The control displays an error message in the header until it is cleared or replaced by a higher-priority error (higher error class). Information that appears only briefly is always displayed.

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

If a rare **processor check error** should occur, the control automatically opens the error window. You cannot correct such an error. Shut down the system and restart the control.

Opening the error window

ERR

- ▶ Press the **ERR** key
- > The control opens the error window and displays all accumulated error messages.

Closing the error window

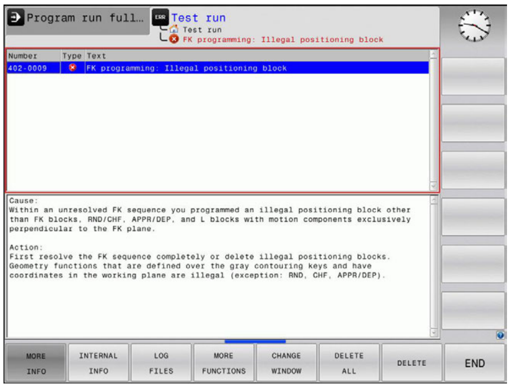
END

- ▶ Press the **END** soft key; or

ERR

- ▶ Press the **ERR** key
- > The control closes the error window.

Detailed error messages



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

- Open the error window

- Information on the error cause and corrective action: Position the cursor on the error message and press the **MORE INFO** soft key
- > The control opens a window with information on the error cause and corrective action.
- Leave Info: Press the **MORE INFO** soft key again

INTERNAL INFO soft key

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

- Open the error window

- Detailed information about the error message: Position the cursor on the error message and press the **INTERNAL INFO** soft key
- > The control opens a window with internal information about the error.
- To exit Details, press the **INTERNAL INFO** soft key again

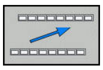
FILTER soft key

The **FILTER** soft key enables you to filter identical warnings listed immediately in succession.

- Open the error window

- Press the **MORE FUNCTIONS** soft key

- Press the **FILTER** soft key The control filters the identical warnings



- Leave Filter: Press the **GO BACK** soft key

Clearing errors

Clearing errors outside of the error window

CE

- Clear the errors/messages in the header: Press the **CE** key

> In certain situations you cannot use the **CE** key for clearing the errors because the key is used for other functions.

Clearing errors

- Open the error window

DELETE

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

DELETE ALL

- Clear all error messages: Press the **DELETE ALL** soft key.

> If the cause of the error has not been corrected, the error message cannot be cleared. In this case, the error message remains in the window.

Error log

The control stores errors occurred and important events (e.g. system start) in an error log. The capacity of the error log is limited. If the log is full, the control uses a second file. If this is also full, the first error log is deleted and newly written etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history.

- Open the error window.

LOG FILES

- Press the **LOG FILES** soft key

ERROR LOG

- Open the error log file: Press the **ERROR LOG** soft key

PREVIOUS FILE

- Set the previous error log if required: Press the **PREVIOUS FILE** soft key

CURRENT FILE

- Set the current error log if required: Press the **CURRENT FILE** soft key

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

Keystroke log

The control stores each key pressed and important events (e.g. system start) in a keystroke log. The capacity of the keystroke log is limited. If the keystroke log is full, the control switches to a second keystroke log. If this is also full, the first keystroke log is deleted and newly written etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history of the inputs.

LOG FILES

- Press the **LOG FILES** soft key

KEYSTROKE LOG

- Open the keystroke log file: Press the **KEYSTROKE LOG** soft key

PREVIOUS FILE

- Set the previous keystroke log if required: Press the **PREVIOUS FILE** soft key

CURRENT FILE

- Set the current keystroke log if required: Press the **CURRENT FILE** soft key

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

Overview of the keys and soft keys for viewing the log

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

Informational texts

If an operating error occurred, e.g. pressing an impermissible key or entering a value outside of a validity range, the control displays an information text in the header to inform you of the operating error. The control deletes this information text with the next valid entry.

Saving service files

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

If you repeat the **SAVE SERVICE FILES** function with the same file name, the previously saved group of service files is overwritten. Therefore, use another file name when executing the function another time.

Saving service files

- Open the error window

LOG FILES

- Press the **LOG FILES** soft key

SAVE SERVICE FILES

- Press the **SAVE SERVICE FILES** soft key
- > The control opens a pop-up window in which you can enter a file name or a complete path for the service file.

OK

- Save the service files: Press the **OK** soft key

Calling the TNCguide help system

You can call the control'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.

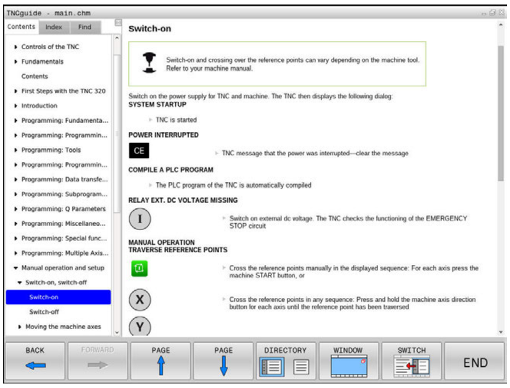
> Refer to your machine manual.
>
> If your machine manufacturer also provides a help system, the control shows an additional **Machine manufacturer (OEM)** soft key with which you can call this separate help system. There you will find further, more detailed information on the error message concerned.

4.10 TNCguide context-sensitive help system

Application

Before you can use the TNCguide, you need to download the help files from the HEIDENHAIN home page.

Further information: "Downloading current help files", page 180



The **TNCguide** context-sensitive help system contains the user documentation in HTML format. The TNCguide is called with the **HELP** key, and the control often immediately displays the information specific to the condition from which the help was called (context-sensitive call). Even if you are editing an NC block and press the **HELP** key, you are usually brought to the exact place in the documentation that describes the corresponding function.

The control tries to start the TNCguide in the language that you have selected as the conversational language. If the required language version is not available, the control automatically opens the English version.

The following user documentation is available in TNCguide:

- Conversational Programming User's Manual (**BHBKlartext.chm**)
- List of All Error Messages **(errors.chm)**

In addition, the **main.chm** "book" file is available, with the contents of all existing .chm files.

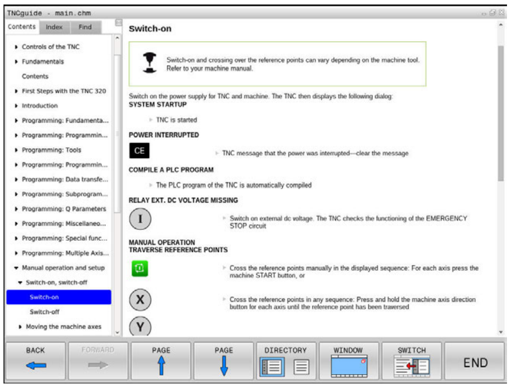
As an option, your machine tool builder can embed machine-specific documentation in the **TNCguide**. These documents then appear as a separate book in the **main.chm** file.

Working with TNCguide

Calling TNCguide

There are several ways to start the TNCguide:

- Press the **HELP** key.
- Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys
- Open a help file (CHM file) via the file management. The control can open any .chm file, even if it is not saved in the control’s internal memory



> On the Windows programming station, the TNCguide is opened in the internally defined standard browser.

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 control 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
- > The control opens the TNCguide. If there is no entry point for the selected soft key, then the control opens the book file **main.chm**. You can search for the desired explanation using full text search or by using the navigation.

Even if you are editing an NC block, context-sensitive help is available:

- Select any NC block
- Select the desired word
- Press the **HELP** key.
- > The control opens the Help system and shows the description of the active function. This does not apply for miscellaneous functions or cycles from your machine manufacturer.

Navigating in the TNCguide

It's easiest to use the mouse to navigate in TNCguide. A table of contents appears on the left side of the screen. By clicking the rightward 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 TNCguide through keys and soft keys. The following table contains an overview of the corresponding key functions.

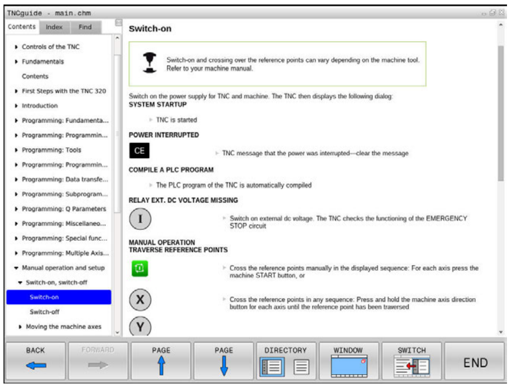
Soft key	Function
↑ ↓	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
→	■ If the table of contents at left is active: Open up the table of contents ■ If the text window at right is active: No function
←	■ If the table of contents at left is active: Close the table of contents ■ If the text window at right is active: No function
ENT	■ 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 next link
BACK	Select the page last shown
FORWARD	Page forward if you have used the **Select page last shown** function
PAGE ↑	Move up by one page
PAGE ↓	Move down by one page

Soft key	Function
DIRECTORY	Display or hide table of contents
WINDOW	Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the control window
SWITCH	The focus is switched internally to the control application so that you can operate the control when the TNCguide is open. If the full screen is active, the control reduces the window size automatically before the change of focus
END	Exit TNCguide

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 arrow keys.

The left side is active.



- Select the **Index** tab
- Use the arrow keys or the mouse to select the desired keyword

Alternative:

- Enter the first few characters
- > The control synchronizes the subject index and creates a list in which you can find the subject more easily.
- Use the **ENT** key to call the information on the selected keyword

> **i** You can enter the search word only with a keyboard connected via USB.

Full-text search

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

The left side is active.

- Select the **Find** tab
- Activate the **Find:** entry field
- Enter the search word
- Press the **ENT** key
- > The control lists all sources containing the word.
- Use the arrow keys to navigate to 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, the control searches only through headings and ignores the body text. To activate the function, use the mouse or select it and then press the space bar to confirm.

You can enter the search word only with a keyboard connected via USB.

Downloading current help files

You'll find the help files for your control software on the HEIDENHAIN homepage:
http://content.heidenhain.de/doku/tnc_guide/html/en/index.html

Navigate to the suitable help file as follows:

- TNC Controls
- Series, e.g. TNC 100
- Desired NC software number, e.g. TNC 128 (77184x-04)
- Select the desired language version from the **TNCguide online help** table
- Download the ZIP file
- Extract the ZIP file
- Move the extracted CHM files to the **TNC:\tncguide\en** directory or the respective language subdirectory on the control

> When using TNCremo to transfer the .chm files to the control, select the binary mode for files with 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	**TNC:\tncguide\sl**
Norwegian	**TNC:\tncguide\no**
Slovak	**TNC:\tncguide\sk**
Korean	**TNC:\tncguide\kr**
Turkish	**TNC:\tncguide\tr**
Romanian	**TNC:\tncguide\ro**

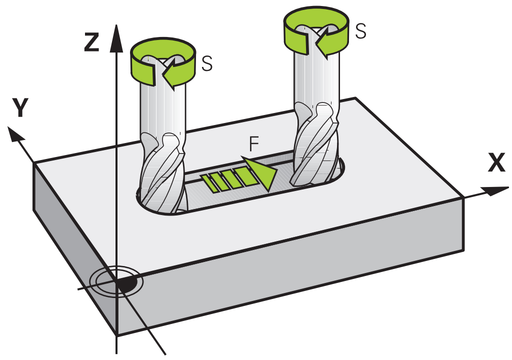
5

Tools

5.1 Entering tool-related data

Feed rate F

The feed rate **F** is the speed 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 **TOOL CALL** block and in every positioning block.

You enter the feed rate **F** in mm/min in millimeter programs, and in 1/10 inch/min in inch-programs, for resolution reasons. Alternatively, with the corresponding soft keys, you can also define the feed rate in mm per revolution (mm/1) **FU** or in mm per tooth (mm/tooth) **FZ**.

Rapid traverse

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

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

Duration of effect

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

Changing during program run

You can adjust the feed rate during the program run with the feed rate potentiometer F.

The feed rate potentiometer lowers the programmed feed rate, not the feed rate calculated by the control.

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in a **TOOL CALL** block (tool call). Instead, you can also define the cutting speed Vc in meters per minute (m/min).

Programmed change

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

TOOL CALL

- Program a tool call: Press the **TOOL CALL** key
- Ignore the dialog question for **Tool number ?** with the **NO ENT** key
- Ignore the dialog question for **Working spindle axis X/Y/Z ?** with the **NO ENT** key
- Enter the new spindle speed for the dialog question **Spindle speed S= ?,** and confirm with **END,** or switch via the **VC** soft key to entry of the cutting speed.

> If the number of the already inserted tool is entered in the **TOOL CALL** block without specifying the tool axis, then only the spindle speed will change.
>
> If the tool axis is also entered in the **TOOL CALL** block, the control will insert a replacement tool if a replacement tool was defined.

Changing during program run

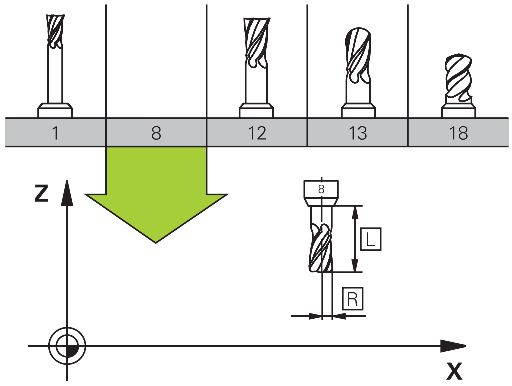
You can adjust the spindle speed during program run with the spindle speed potentiometer S.

5.2 Tool data

Requirements for tool compensation

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

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



Tool number, tool name

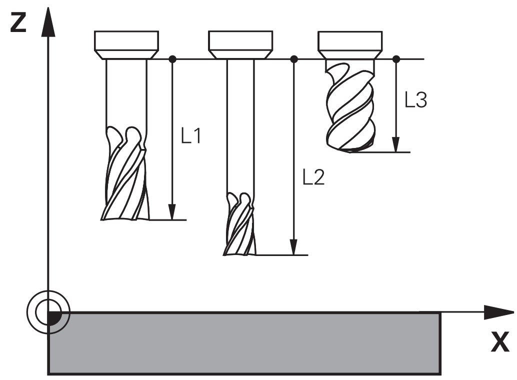
Each tool is identified by a number between 0 and 32767. If you are working with tool tables, you can also enter a tool name for each tool. Tool names can have up to 32 characters.

> **Permitted special characters**: # $ % & , - _ . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
>
> The control automatically replaces lowercase letters with corresponding uppercase letters during saving.
>
> **Impermissible characters**: <blank space> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~

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

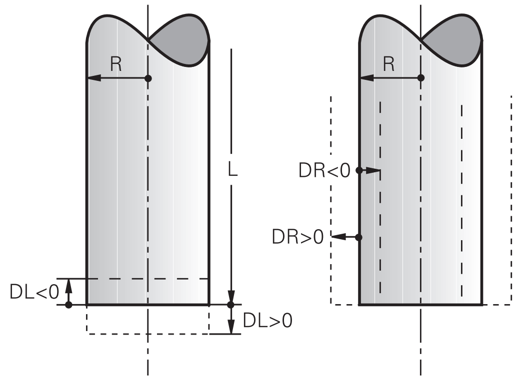
You should always enter the tool length L as an absolute value based on the tool reference point.



Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii



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

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

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

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

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

> Delta values from the tool table influence the graphical representation of the clearing simulation.
>
> Delta values from the **TOOL CALL** block do not change the represented size of the **tool** during the simulation. However, the programmed delta values move the **tool** by the defined value in the simulation.

Entering tool data into the NC program

> Refer to your machine manual.
>
> The machine tool builder determines the scope of functions of the **TOOL DEF** function.

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

- Select the 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

Example

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

Entering tool data into the table

You can define and store up to 32 767 tools and their tool data in a tool table. Also see the editing functions later in this chapter.

You must use tool tables if:

- you wish to use indexed tools such as stepped drills with more than one length compensation value
  Further information: "Indexed tool", page 189
- your machine tool has an automatic tool changer
- you want to work with Cycles 25x

NOTICE

Caution: Data may be lost!

Deleting line 0 from the tool table will destroy the structure of the table. As a result, locked tools might no longer be recognized as locked and, consequently, the search for a replacement tool will not work, either. The problem cannot be solved by reinserting a line 0. The original tool table will be permanently damaged!

- ▶ Restore the tool table
  - Add a new line 0 to the defective tool table
  - Copy the defective tool table (e.g. toolcopy.t)
  - Delete the defective tool table (current tool.t)
  - Copy the copied tool table (toolcopy.t) as tool.t
  - Delete the copied tool table (toolcopy.t)
- ▶ Contact HEIDENHAIN Service (NC helpline)

All table names must start with a letter. Please keep this in mind when creating and managing additional tables.

You can select the table view with the **Screen Layout** key. You can choose between a list view and a form view.

Indexed tool

Step drills, T-slot milling cutters, side milling cutters and, in generally, all tools that require the input of multiple length and radius data cannot be fully defined in a single line of the tool table. Each line of the table permits the definition of one length and one radius.

In order to assign multiple compensation data to a tool (multiple tool table lines), add an indexed tool number (such as **T 5.1**) to an existing tool definition (**T 5**). Each additional line of the table thus comprises the original tool number, a period and an index (in ascending order from 1 to 9). The original tool table line contains the maximum tool length; the tool lengths in the subsequent table lines are given in descending order of their distance to the tool holder point.

Proceed as follows to create an indexed tool number (table line):

- ▶ Open the tool table
- ▶ Press the **Insert Line** soft key
- > The control opens the **Insert Line** pop-up window
- ▶ In the **Number of new lines =** input field, enter the number of lines to add
- ▶ Enter the original tool number into the **Tool number** input field
- ▶ Confirm with **OK**
- > The control adds the additional lines to the tool table

Quick search for the tool name:

If the **EDIT** soft key is set to **OFF**, you can search for a tool name. Proceed as follows:

- ▶ Enter the first few characters of the tool name, e.g. **MI**
- > The control shows a dialog box with the entered text and jumps to the first match.
- ▶ Enter additional characters to narrow down the search result, e.g. **MILL**
- ▶ If the control cannot find any more matches for the entered search string, you can press the last entered character (e.g. **L**) to jump between matches, as with the arrow keys.

The quick search can also be used for tool selection in the **TOOL CALL** block.

Tool table: Standard tool data

Abbr.	Inputs	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 (max. 32 characters, all capitals, no spaces)	**Tool name?**
L	Tool length L	**Tool length?**
R	Tool radius R	**Tool radius?**
R2	Tool radius R2 for toroid cutters (only for graphical representation of a machining operation with spherical or toroid cutters)	**Tool radius 2?**
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 2?**
TL	Set tool lock (**TL** for **T**ool **L**ocked	**Tool locked? Yes=ENT/ No=NOENT**
RT	Number of a replacement tool – if available – as replacement tool (**RT**: for **R**eplacement **T**ool) An empty field or input **0** means no replacement tool has been defined.	**Replacement tool?**
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information	**Maximum tool age?**
TIME2	Maximum tool life in minutes during a tool call: If the current tool age reaches or exceeds this value, the control inserts the replacement tool during the next **TOOL CALL** (if the tool axis is specified)	**Max. tool age for TOOL CALL?**
CUR_TIME	Current age of the tool in minutes: The control automatically counts the current tool life (**CUR_TIME**: For **CUR**rent **TIME**) A starting value can be entered for used tools	**Current tool age?**

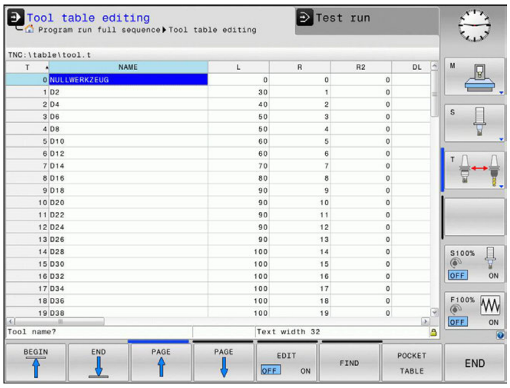
Abbr.	Inputs	Dialog
TYPE	Tool type: Press the **ENT** key to edit the field. The **GOTO** key opens a window for selecting the tool type (in the tool management, press the **SELECT** soft key to open a pop-up window). You can assign tool types to specify the display filter settings such that only the selected type is visible in the table	**Tool type?**
DOC	Comment on tool (max. 32 characters)	**Tool description**
PLC	Information on this tool that is to be sent to the PLC	**PLC status?**
LCUTS	Tooth length of the tool	**Tooth length in the tool axis?**
NMAX	Limit the spindle speed for this tool. The programmed value is monitored (error message) as well as an increase in the shaft speed via the potentiometer. Function inactive: Enter **-**. **Input range**: 0 to +999 999 if function not active: enter **-**	**Maximum speed [rpm]**
TP_NO	Reference to the number of the touch probe in the touch-probe table	**Number of the touch probe**
T-ANGLE	Point angle of the tool.	**Point angle**
PITCH	Thread pitch of the tool. Used by tapping cycles (Cycle 206, Cycle 207 and Cycle 209). A positive algebraic sign means a right-hand thread.	**Tool thread pitch?**
LAST_USE	Date and time that the tool was last inserted via **TOOL CALL**	**Date/time of last tool call**
KINEMATIC	Press the **SELECT** soft key to display the tool carrier kinematics (in the tool management, press the **SELECT** soft key) and press the **OK** soft key to confirm the file name and path. **Further information:** "Allocating parameterized tool carriers", page 372	**Tool-carrier kinematics**
OVRTIME	Time for exceeding the tool life in minutes **Further information:** "Overtime for tool life", page 204 Function is defined by the machine manufacturer. Refer to your machine manual.	**Tool life expired**

Tool table: Tool data required for automatic tool measurement

> Refer to your machine manual.
> The machine tool builder defines whether the **R-OFFS** offset will be taken into account for a tool with **CUT** 0.

Abbr.	Inputs	Dialog
CUT	Number of teeth (99 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control 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 control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT	Cutting direction of the tool for measuring the tool during rotation	**Cutting direction? M4=ENT/ M3=NOENT**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L-OFFS	Tool radius measurement: tool offset in addition to **offsetToolAxis** 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 control locks the tool (status **L**). Input range: 0 to 3.2767 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Editing the tool table



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

Other tool tables that are to be archived or used for test runs are given different file names with the extension .T. By default, for the **Test Run** and **Programming** modes the control also uses the TOOL.T tool table. In the **Test Run** mode, press the **TOOL TABLE** soft key to edit it.

To open the tool table TOOL.T:

- Select any machine operating mode

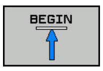
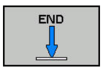
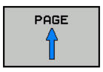
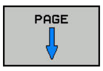
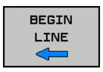
- Select the tool table: Press the **TOOL TABLE** soft key

- Set the **EDIT** soft key to **ON**

> If you edit the tool table, the selected tool is locked. If this tool is required in the NC program being used, the control shows the message: **Tool table locked**.
>
> If a new tool is created, the length and radius columns remain empty until entered manually. An attempt to insert such a newly created tool will be aborted by the control and an error message will appear. This means you cannot insert a tool for which no geometry data are available yet.

Proceed as follows to use the keyboard or a connected mouse for navigation and editing:

- Arrow keys: move from one cell to the next
- ENT key: jump to the next cell; with selection fields: open the selection dialog
- Mouse click on a cell: move to the cell
- Double click on a cell: place the cursor in the cell; with selection fields: open the selection dialog

Soft key	Editing functions of the tool table
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Find the text or number
	Go to beginning of line

Soft key	Editing functions of the tool table
END LINE	Go to end of line
COPY FIELD	Copy active field
PASTE FIELD	Insert copied field
APPEND N LINES	Add the entered number of lines (tools) at the end of the table
INSERT LINE	Insert a line with definable tool number
DELETE LINE	Delete the current line (tool)
SORT	Sort the tools according to the content of a column
SELECT	Select possible entries from a pop-up window
RESET COLUMN	Reset the value
EDIT CURRENT FIELD	Place the cursor in the current cell

Displaying only specific tool types (filter setting)

- Press the **TABLE FILTER** soft key
- Select the desired tool type by soft key
- > The control displays only tools of the selected type.
- Cancel the filter: Press the **SHOW ALL** soft key

> Refer to your machine manual.
> The machine tool builder adapts the features of the filter function to the requirements of your machine.

Soft key	Filter functions of the tool table
TABLE FILTER	Select the filter function
SHOW ALL	Cancel the filter settings and show all tools
DEFAULT FILTER	Use the default filter
DRILL	Show all drills in the tool table
CUTTER	Show all cutters in the tool table
THREADTOOL	Show all taps/thread cutters in the tool table
TCH. PROBE	Show all touch probes in the tool table

Hiding or sorting the tool table columns

You can adapt the layout of the tool table to your needs. Columns that are not to be displayed can be simply hidden:

- Press the **HIDE/ SORT/ COLUMNS** soft key
- Select the appropriate column name with the arrow key
- Press the **HIDE COLUMN** soft key to remove this column from the table view

You can also modify the sequence of columns in the table:

- You can also modify the sequence of columns in the table with the **Move before:** dialog. The entry highlighted in **Displayed columns:** is moved in front of this column

Use a connected mouse or the control's keyboard to navigate in the form. Navigation using the control's keyboard:

- Press the navigation keys to go to the input fields.
- Use the arrow keys to navigate within an input field.
- To open pop-down menus, press the **GOTO** key.

> The function **freeze number of columns** enables you to determine how many columns (0-3) the control will freeze to the left border of the screen. These columns will remain visible when you navigate to the right within the table.

Opening any other tool table

- Select the **Programming** operating mode

PGM MGT

- To call the file manager, press the PGM MGT key
- Select a file or enter a new file name. Confirm 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.

Further information: "Editing the tool table", page 193

Exiting any other tool table

- Call the file manager and select a file of a different type, such as an NC program

Importing tool tables

> Refer to your machine manual.
>
> The machine tool builder can adapt the **ADAPT NC PGM / TABLE** function.
>
> The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs.

If you export a tool table from an iTNC 530 and import it into a TNC 128, you have to adapt its format and content before you can use the tool table. On the TNC 128, you can adapt the tool table conveniently with the **ADAPT NC PGM / TABLE** function. The control converts the contents of the imported tool table to a format valid for the TNC 128 and saves the changes to the selected file.

Follow this procedure:

- ▶ Save the tool table of the iTNC 530 to the **TNC:\table** directory
  - ▶ Select the **Programming** operating mode

PGM MGT
  - ▶ Press the **PGM MGT** key
  - ▶ Move the cursor to the tool table you want to import

MORE FUNCTIONS
  - ▶ Press the **MORE FUNCTIONS** soft key

ADAPT NC PGM / TABLE
  - ▶ Press the **ADAPT NC PGM / TABLE** soft key
  - > The control asks you whether you want to overwrite the selected tool table.
  - ▶ Press the **CANCEL** soft key
  - ▶ Alternative: Press the **OK** soft key to overwrite
- ▶ Open the converted table and check its contents
- > New columns in the tool table are highlighted green
- ▶ Press the **REMOVE UPDATE INFORMATION** soft key
- > The green columns are displayed in white again

> The following characters are permitted in the **Name** column of the tool table: # $ % & , - . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z _
>
> During the import, a comma is converted to a period.
>
> The control overwrites the active tool table when importing an external table with the same name. To prevent data loss, back up the original tool table before you start the import!
>
> When iTNC 530 tool tables are imported, all defined tool types are transferred as well. Tool types not present are imported as type **Undefined**. Check the tool table after the import.

Overwriting tool data from an external PC

Application

The HEIDENHAIN data transfer software TNCremo provides an especially convenient way to use an external PC to overwrite tool data.

Further information: "Software for data transfer", page 483

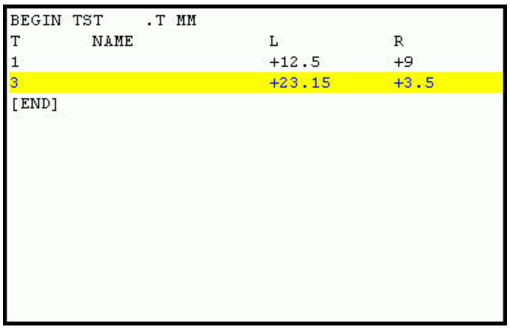
This application case occurs if you wish to determine tool data on an external tool presetter and then transfer this to the control.

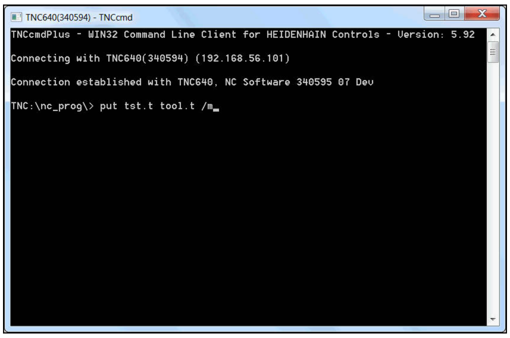
Requirements

In addition to option 18 HEIDENHAIN DNC, TNCremo (from version 3.1) is required with TNCremoPlus functions.

Procedure

- Copy the TOOL.T tool table to the control, for example to TST.T
- Start the data transfer software TNCremo on the PC
- Connect to the control
- 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 number (column T) need not be consecutive
- In TNCremo, select the menu item <Extras> and <TNCcmd>: This starts TNCcmd.
- To transfer the TST.T file to the control, enter the following command and confirm with the return key (see figure):
  put tst.t tool.t /m

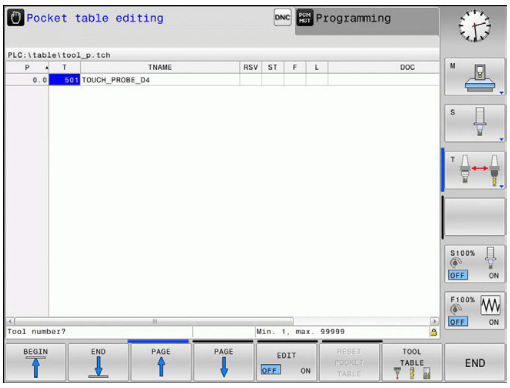




> During transfer, only the tool data defined in the subfile (e.g. TST.T) is overwritten. All other tool data of the table TOOL.T remains unchanged.
>
> The procedure for copying tool tables using the file manager is described in the file management.
>
> **Further information:** "Copying a table", page 135

Pocket table for tool changer

> Refer to your machine manual.
> The machine tool builder adapts the features of the pocket table to the requirements of your machine.



For automatic tool changing you need the a pocket table. You manage the assignment of your tool changer in the pocket table. The pocket table is in the **TNC:\table** directory. The machine manufacturer can amend the name, path and content of the pocket table. If you wish, you can select different views using the soft keys in the **TABLE FILTER** menu.

Editing a pocket table in a Program Run operating mode

- Select the tool table: Press the **TOOL TABLE** soft key

- 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 Programming mode

Proceed as follows to select the pocket table in the Programming mode of operation:

PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Press the **SHOW ALL** soft key
- Select a file or enter a new file name
- Confirm your entry with the **ENT** key or the **SELECT** soft key

Abbr.	Inputs	Dialog
P	Pocket number of the tool in the tool magazine	-
T	Tool number	**Tool number?**
RSV	Pocket reservation for box magazines	**Pocket reserv.: Yes = ENT / No = NOENT**
ST	Special tool (**ST**); If your special tool blocks pockets in front of and behind its actual pocket, these additional pockets need to be locked in column L (status L).	**Special tool?**
F	The tool is always returned to the same pocket in the tool magazine	**Fixed pocket? Yes = ENT / No = NO ENT**
L	Locked pocket (**L**: for **L**ocked)	**Pocket locked Yes = ENT / No = NO ENT**
DOC	Display of the comment to the tool from TOOL.T	-
PLC	Information on this tool pocket that is to be sent to the PLC	**PLC status?**
P1 ... P5	Function is defined by the machine tool builder. The machine tool documentation provides further information	**Value?**
PTYP	Tool type. Function is defined by the machine tool builder. The machine tool documentation provides further information	**Tool type for pocket table?**
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?**

Soft key	Editing functions for pocket tables
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
RESET POCKET TABLE	Reset pocket table Depends on optional machine parameter **enableReset** (no.106102)
RESET COLUMN T	Reset tool number T column Depends on machine parameter **showResetColumnT** (no.)
BEGIN LINE	Go to beginning of line
END LINE	Go to end of line
SIMULATED TOOL CHANGE	Simulate a tool change
SELECT	Select a tool from the tool table: The control shows the contents of the tool table. Use the arrow keys to select a tool, press **OK** to transfer it to the pocket table
RESET COLUMN	Reset the value
EDIT CURRENT FIELD	Place the cursor in the current cell
SORT	Sort the view

Refer to your machine manual.

The machine manufacturer defines the features, properties and designations of the various display filters.

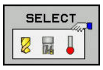
Calling the tool data

Before you can call the tool, you have to define it in a **TOOL DEF** block or in the tool table.

A **TOOL CALL** in the NC program is programmed with the following data:

TOOL CALL

- Press the **TOOL CALL** key
- **Tool number**: Enter the number or name of the tool. With the **TOOL NAME** soft key you can enter a name. With the **QS** soft key you enter a string parameter. The control automatically places the tool name in quotation marks. You have to assign a tool name to a string parameter first. Names always refer to an entry in the active tool table TOOL .T.



- Alternative: Press the **SELECT** soft key
- > The control opens a window where you can select a tool directly from the TOOL.T tool table.
- To call a tool with other compensation values, enter a decimal point followed by the index you defined in the tool table.
- **Working spindle axis X/Y/Z**: Enter the tool axis
- **Spindle speed S**: Enter the spindle speed S in revolutions per minute (rpm) Alternatively, you can define the cutting speed Vc in meters per minute (m/min). Press the **VC** soft key
- **Feed rate F**: Enter feed rate **F** in millimeters per minute (mm/min). Alternatively, you can define the feed rate in millimeters per revolution (mm/1) by pressing the **FU** soft key or in millimeters per tooth (mm/tooth) by pressing **FZ**. The feed rate is effective until you program a new feed rate in a positioning block or in a **TOOL CALL** block
- **Tool length oversize DL**: Enter the delta value for the tool length
- **Tool radius oversize DR**: Enter the delta value for the tool radius
- **Tool radius oversize DR2**: Enter the delta value for the tool radius 2

> If the number of the already inserted tool is entered in the **TOOL CALL** block without specifying the tool axis, then only the spindle speed will change.
>
> If the tool axis is also entered in the **TOOL CALL** block, the control will insert a replacement tool if a replacement tool was defined.

Tool selection in the pop-up window

You can search for a tool in the pop-up window:

GOTO □

- ▶ Press the **GOTO** key
- ▶ Alternative: Press the **FIND** soft key
- ▶ Enter the tool name or tool number

ENT

- ▶ Press the **ENT** key
- > The control goes to the first tool that matches the entered search string.

The following functions can be used with a connected mouse:

- You can sort the data in ascending or descending order by clicking a column of the table head.
- You can arrange the columns in any sequence you want by clicking a column of the table head and then moving it with the mouse key pressed down

The pop-up windows displayed for a tool number search and a tool name search can be configured separately. The sort order and the column widths are retained when the control is switched off.

Tool call

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

Example

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

The character **D** preceding **L**, **R** and **R2** designates delta values.

Preselection of tools

> Refer to your machine manual.
> The preselection of tools with **TOOL DEF** can vary depending on the individual machine tool.

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

Tool change

Automatic tool change

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

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

Overtime for tool life

> This feature must be enabled and adapted by the machine tool builder.

The tool condition at the end of planned tool life depends on e.g. the tool type, machining method and workpiece material. In the **OVRTIME** column of the tool table, enter the time in minutes for which the tool is permitted to be used beyond the tool life.

The machine manufacturer specifies whether this column is enabled and how it is used during tool search.

Tool usage test

Requirements

Refer to your machine manual.

The tool usage test function must be enabled by your machine tool builder.

To conduct a tool usage test, you must activate **Create tool usage files** in the MOD menu.

Further information: "Tool usage file", page 472

Generating a tool usage file

Depending on the setting in the MOD menu, you have the following options for generating the tool usage file:

- Completely simulate the NC program in the **Test Run** operating mode
- Completely run the NC program in the **Program Run, Full Sequence/Single Block** operating modes
- In the **Test Run** operating mode, press the **GENERATE TOOL USAGE FILE** soft key (also possible without simulation)

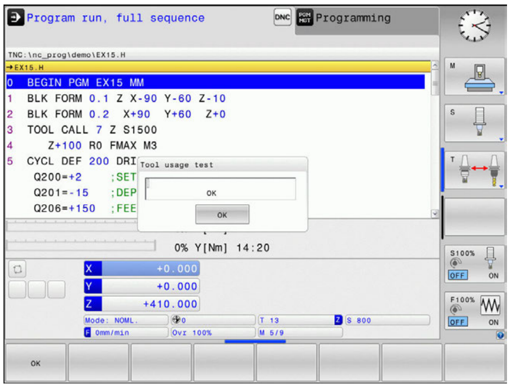
The tool usage file generated is in the same directory as the NC program. It contains the following information:

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. 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 control saves the path name of the corresponding NC program. The **TIME** column shows the sum of all **TIME** entries (feed time without rapid traverse movements). The control sets all other columns to 0 ■ **TOOLFILE**: In the **PATH** column, the control saves the path name of the tool table with which you conducted the test run. This enables the control during the actual tool usage test to detect whether you performed the test run with TOOL.T
TNR	Tool number (**-1**: Tool not inserted yet)
IDX	Tool index
NAME	Tool name from the tool table
TIME	Tool usage time in seconds (feed time without rapid traverse movements)
WTIME	Tool-usage time in seconds (total usage time between tool changes)
RAD	**Tool radius R** + **Oversize of tool radius DR** from the tool table. (in mm)
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
T	Tool number with tool index
OVRMAX	Maximum feed rate override that occurred during machining. The control enters the value 100 (%) during the test run
OVRMIN	Minimum feed rate override that occurred during machining. The control enters the value -1 during the test run
NAMEPROG	■ 0: The tool number is programmed ■ 1: The tool name is programmed

The control saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. This file is not visible unless the machine parameter **dependentFiles** (no. 122101) is set to **MANUAL**

Using a tool usage test

Before starting a program in the **Program Run, Full Sequence/ Single Block** operating modes, you can check whether the tools being used in the selected program are available and have sufficient remaining service life. The control then compares the actual service-life values in the tool table with the nominal values from the tool usage file.



TOOL USAGE

- Press the **TOOL USAGE** soft key

TOOL USAGE TEST

- Press the **TOOL USAGE TEST** soft key
- > The control opens the **Tool usage test** pop-up window indicating the result of the usage test.

OK

- Press the **OK** soft key
- > The control closes the pop-up window.

ENT

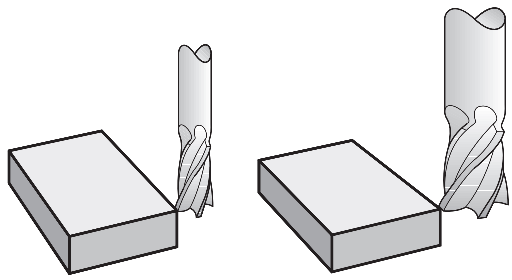
- Alternative: Press the **ENT** key

You can query the tool usage test with the **FN 18 ID975 NR1** function.

5.3 Tool compensation

Introduction

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



Tool length compensation

Length compensation becomes effective automatically as soon as a tool is called. To cancel length compensation, call a tool with the length L=0 (e.g. **TOOL CALL 0**).

NOTICE

Danger of collision!

The control uses the defined tool lengths for tool length compensation. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform a length compensation and a collision check for tools with a length of **0** and after **TOOL CALL 0**. Danger of collision during subsequent tool positioning movements!

- Always define the actual tool length of a tool (not just the difference)
- Use **TOOL CALL 0** only to empty the spindle

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

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

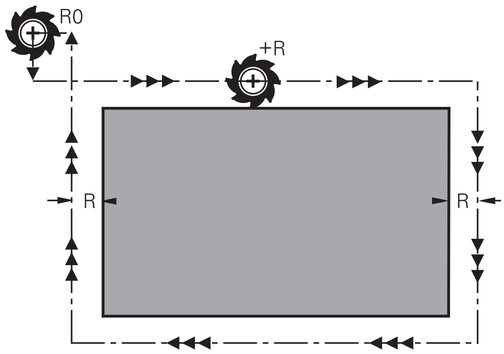
L:	Tool length **L** from **TOOL DEF** block or tool table
$\mathbf{DL}_{\text{TOOL CALL}}$:	Oversize for length **DL** in the **TOOL CALL** block
$\mathbf{DL}_{\text{TAB}}$:	Oversize for length **DL** in the tool table

Tool radius compensation with paraxial positioning blocks

The control can compensate the tool radius in the working plane with the aid of paraxial positioning blocks. You can enter the dimensions directly from the drawing without first having to convert the positions. The TNC extends or shortens the traverse path by the tool radius.

- **R+** extends the tool path by the tool radius
- **R-** shortens the tool path by the tool radius
- **R0** positions the tool using the tool center

The radius compensation is effective as soon as a tool is called and traversed with a paraxial movement in the working plane with **R+**/**R-**.



> Radius compensation is not effective for positioning movements in the spindle axis.
>
> The last selected radius compensation remains active in a positioning block that does not contain any information about radius compensation.

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

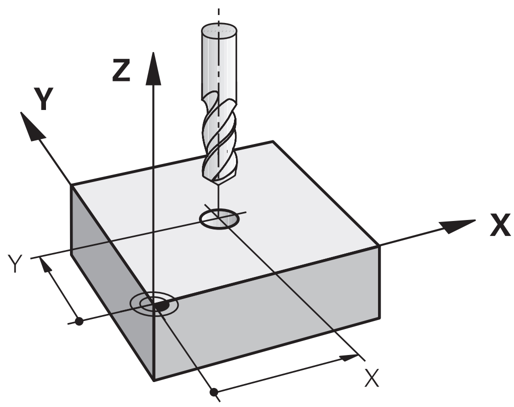
Compensation value = **R** + **DR**$_{\text{TOOL CALL}}$ + **DR**$_{\text{TAB}}$ with

R:	Tool radius **R** from **TOOL DEF** block or tool table
DR $_{\text{TOOL CALL}}$:	Oversize for radius **DR** in the **TOOL CALL** block
DR $_{\text{TAB}}$:	Oversize for radius **DR** in the tool table

Contouring without radius compensation: R0

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

Applications: Drilling and boring, pre-positioning



Entering radius compensation

Radius compensation is entered in a positioning block. Enter the coordinates of the target point and confirm your entry with the **ENT** key.

TOOL RADIUS COMP: R+/R-/NO COMP?

R+
- The TNC extends the traverse path of the tool by the tool radius

R−
- The TNC shortens the traverse path of the tool by the tool radius

ENT
- Select tool movement without radius compensation or cancel radius compensation: Press the **ENT** key

END
- Terminate the block: Press the **END** key

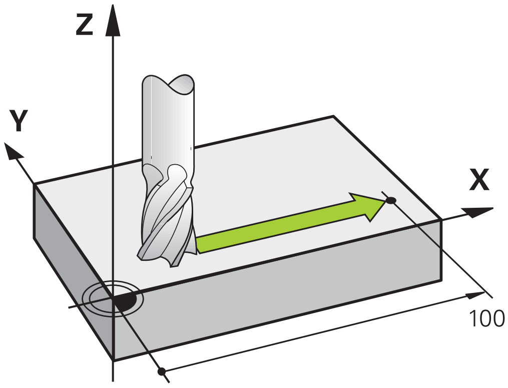
6

Programming tool - movements

6.1 Fundamentals

Structure blocks in NC program

The orange axis keys initiate the dialog for a paraxial positioning block. The control asks you successively for all the necessary information and inserts the program block into the NC program.



- **Coordinates** of the end point of the movement
- **Radius compensation R+/R-/R0**
- **Feed rate F**
- **Miscellaneous function M**

Example NC block

```
6  X+45 R+ F200 M3
```

You always program the direction of tool movement. 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.

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning can also lead to contour damage. There is danger of collision during the approach movement!

- Program a suitable pre-position
- Check the sequence and contour with the aid of the graphic simulation

Radius compensation

The control can compensate the tool radius automatically. In paraxial positioning blocks, you can select whether the control lengthens the traverse by the tool radius (R +) or shortens it (R-).

Further information: "Tool radius compensation with paraxial positioning blocks", page 209

Miscellaneous functions M

With the control's miscellaneous functions you can affect

- the program run, e.g., a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off

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. In addition, you can have a part program call a separate program for execution.

Further information: "Subprograms and Program Section Repeats", page 221

Programming with Q parameters

Instead of programming numerical values in a machining 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, programming with Q parameters enables you to measure with the 3-D touch probe during the program run.

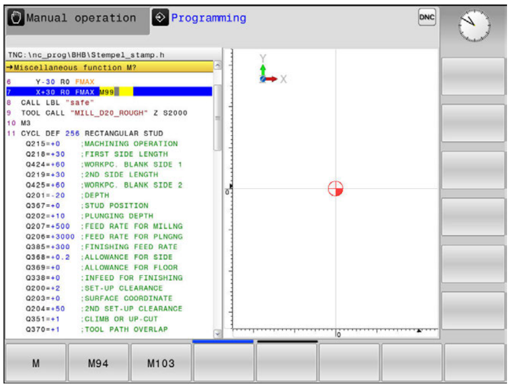
Further information: "Programming Q Parameters", page 241

6.2 Tool movements

Programming tool movements for workpiece machining

Create an NC block with the axis keys

Use the orange axis keys to initiate the dialog. The control asks you successively for all the necessary information and inserts the program block into the NC program.



Example—programming a straight line

X

- Select the axis key you want to use for the positioning movement, e.g. **X**

COORDINATES?

- **10** Enter the coordinate of the end point, e.g. 10

ENT

- Press the **ENT** key

TOOL RADIUS COMP: R+/R-/NO COMP?

R0

- Select radius compensation, e.g. by pressing the **R0** soft key
- \> The tool moves without compensation.

Feed rate F=? / F MAX = ENT

- **100** Enter the feed rate, e.g. 100 mm/min. (For programming in inches: Entry of 100 corresponds to a feed rate of 10 inches/min.)

ENT

- Press the **ENT** key

F MAX

- As an alternative, move at rapid traverse: press the **FMAX** soft key

F AUTO

- As an alternative, traverse with the feed rate defined in the **TOOL CALL** block: Press the **F AUTO** soft key

MISCELLANEOUS FUNCTION M?

- **3** (the miscellaneous function **M3** switches on the spindle)

ENT

- The control ends this dialog with the **ENT** key

The program-block window displays the following line:

```
6 X+10 R0 FMAX M3
```

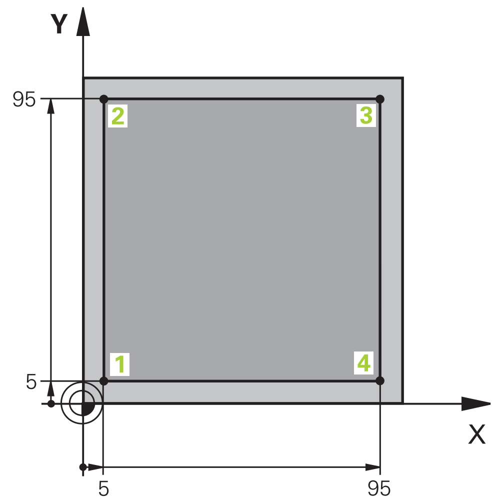
Capture actual position

You can also generate a positioning block with the **actual-position-capture** key:

- In the **Manual operation** mode, move the tool to the position to be captured
- Select the **Programming** operating mode
- Select the NC block after which you want to insert the block

- Press the **Actual-Position-Capture** key
- > The control generates an block.
- Select the desired axis, e.g. by pressing the **ACT. POS. X** soft key
- > The control loads the actual position and ends the dialog.

Example: Linear movement



0 BEGIN PGM LINEAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Call the tool in the spindle axis and with the spindle speed S
4 Z+250 R0 FMAX	Retract the tool in the spindle axis at rapid traverse FMAX
5 X-10 R0 FMAX	Pre-position the tool
6 Y-10 R0 FMAX	Pre-position the tool
7 Z+2 R0 FMAX	Pre-position the tool
8 Z-5 R0 F1000 M13	Move to working depth at feed rate F = 1000 mm/min
9 X+5 R- F500	Contour approach
10 Y+95 R+	Move to point 2
11 X+95 R+	Move to point 3
12 Y+5 R+	Move to point 4
13 X-10 R0	Close the contour and retract
14 Z+250 R0 FMAX M30	Retract the tool, end program
16 END PGM LINEAR MM	

7

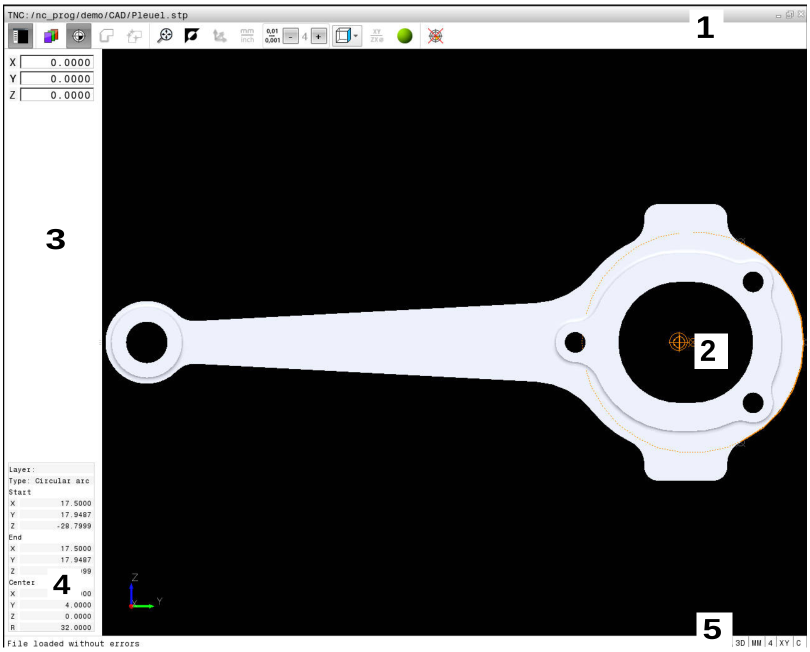
Data Transfer from CAD Files

7.1 Screen layout of the CAD viewer

Fundamentals of the CAD viewer

Screen display

When you open the **CAD-Viewer**, the following screen layout is displayed:



1 Menu bar
2 Graphics window
3 List View window
4 Window element information
5 Status bar

File formats

The **CAD-Viewer** enables you to open standardized CAD data formats directly on the control.

The control displays the following file formats:

File	Type	Format
Step	.STP and .STEP	■ AP 203 ■ AP 214
IGES	.IGS and .IGES	■ Version 5.3
DXF	.DXF	■ R10 to 2015

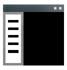
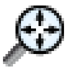
7.2 CAD viewer

Application

The file can simply be selected via the file manager of the control, just like NC programs. This allows you to view models quickly and easily.

The preset can be positioned anywhere in the model. Starting from this preset, element information such as centers of circles can be shown. However, the control cannot execute it.

The following icons are available:

Icon	Setting
	Show or hide the Window List view to expand the Graphics window
	Display of the various layers
 	Set a preset or delete a set preset
	Set the zoom to the largest possible view of the complete graphics
	Change the background color (black or white)
0,01 ... 0,001	Set resolution: The resolution specifies how many decimal places the control will use when generating the contour program. Default setting: 4 decimal places with **mm** and 5 decimal places with **inch**
	Switch between various views of the model e.g. **Top**

> You can use icons to select contours and drilling positions, but the control cannot execute the elements.

8

Subprograms and Program Section Repeats

8.1 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 necessary.

Label

The beginnings of subprograms and program section repeats are marked in a part program by labels **(LBL)**.

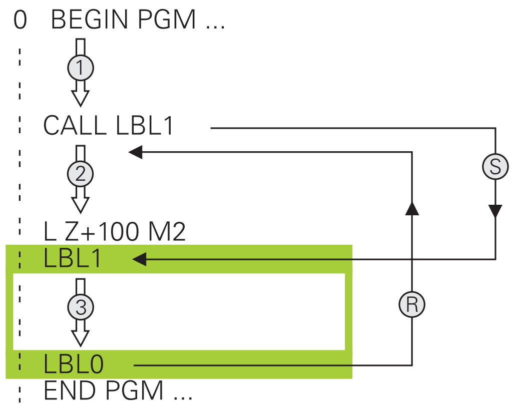
A LABEL is identified by a number between 1 and 65535 or by a name you define. Each LABEL number or LABEL name can be set only once in the program with the **LABEL SET** key. The number of label names you can enter is only limited by the internal memory.

> Do not use a label number or label name more than once!

Label 0 (**LBL 0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

8.2 Subprograms

Operating sequence



1. The control executes the part program up to the **CALL LBL** command for calling a subprogram
2. The subprogram is then executed until the subprogram end **LBL 0**
3. The control then resumes the part program from the block after the subprogram call **CALL LBL**

Programming notes

- A main program can contain any number of subprograms
- You can call subprograms in any sequence and as often as desired
- A subprogram cannot call itself
- Write subprograms after the block with M2 or M30
- If subprograms are located before the block with M2 or M30 in the part program, they will be executed at least once even if they are not called

Programming the subprogram

- To mark the beginning: Press the **LBL SET** key
- Enter the subprogram number. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the text
- Mark the end: Press the **LBL SET** key and enter the label number **0**

Calling a subprogram

LBL CALL

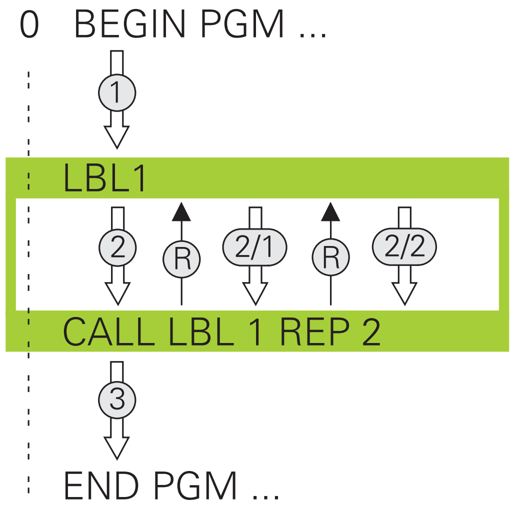
- Call a subprogram: Press the **LBL CALL** key
- Enter the subprogram number of the subprogram you wish to call. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- If you want to enter the number of a string parameter as target address, press the QS soft key
- > The control then jumps to the label name that is specified in the string parameter defined.
- Ignore repeats **REP** by pressing the **NO ENT** key. Repeat **REP** is used only for program section repeats

> **CALL LBL 0** is not permitted (Label 0 is only used to mark the end of a subprogram).

8.3 Program-section repeats

Label

The beginning of a program section repeat is marked by the label **LBL**. The end of a program section repeat is identified by **CALL LBL n REPn**.



Operating sequence

1 The control executes the part program up to the end of the program section (**CALL LBL n REPn**)
2 Then the program section between the called LABEL and the label call **CALL LBL n REPn** is repeated the number of times entered after **REP**
3 The control then continues with the part program

Programming notes

- You can repeat a program section up to 65 534 times in succession
- The total number of times the program section is executed is always one more than the programmed number of repeats, because the first repeat starts after the first machining process.

Programming a program section repeat

LBL SET

- To mark the beginning, press the **LBL SET** key and enter a LABEL NUMBER for the program section you wish to repeat. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the program section

Calling a program section repeat

LBL CALL

- Call a program section: Press the **LBL CALL** key
- Enter the program section number of the program section to be repeated. If you want to use a LABEL name, press the **LBL NAME** soft key to switch to text entry
- Enter the number of repeats **REP** and confirm with the **ENT** key.

8.4 Any desired NC program as subprogram

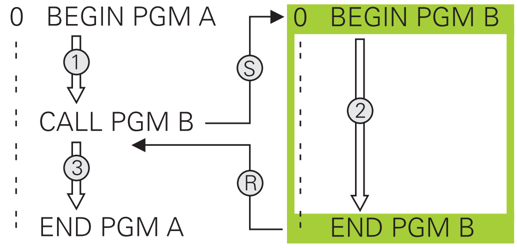
Overview of the soft keys

When you press the **PGM CALL** key, the control displays the following soft keys:

Soft key	Function
CALL PROGRAM	Call an NC program with **PGM CALL**
SELECT DATUM TABLE	Select a datum table with **SEL TABLE**
SELECT POINT TABLE	Select a point table with **SEL PATTERN**
SELECT PROGRAM	Select an NC program with **SEL PGM**
CALL SELECTED PROGRAM	Call the last selected file with **CALL SELECTED PGM**
SELECT CYCLE	Select any NC program with **SEL CYCLE** as a fixed cycle

Operating sequence

1 The control executes the NC program up to the block in which another NC program is called with **CALL PGM**.
2 Then the control executes the called NC program up to the end of program
3 The control then resumes executing the calling NC program with the block after the program call



Programming notes

- The control does not require any labels to call any part program
- The called NC program must not contain any **CALL PGM** call into the calling NC program (an endless loop ensues)
- The called NC program must not contain the miscellaneous functions **M2** or **M30**. If you have defined subprograms with labels in the called NC program, you can then replace M2 or M30 with the **FN 9: If +0 EQU +0 GOTO LBL 99** jump function

If the called NC program contains the miscellaneous functions **M2** or **M30**, then the control displays a warning. The control automatically clears the warning as soon as you select another NC program.

Calling any program as a subprogram

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. If coordinate transformations are not specifically reset in called NC programs, then these transformation are likewise effective for the calling NC program. Danger of collision during machining!

- Reset coordinate transformations used in the same NC program
- Check the machining sequence using a graphic simulation if required

Programming notes:

- If the program you want to call is located in the same directory as the program you are calling it from, then you only need to enter the program name.
- If the program called is not located in the same directory as the calling program, you must enter the complete path, for example **TNC:\ZW35\HERE \PGM1.H**

  Alternatively, you can program relative paths:
  - Starting from the folder of the calling program one folder level up **..\PGM1.H**
  - Starting from the folder of the calling program one folder level down **DOWN\PGM1.H**
  - Starting from the folder of the calling program one folder level up and in one other folder **..\THERE \PGM3.H**
- If you want to call a DIN/ISO program, enter the file type .I after the program name.
- You can also call a program with Cycle **12 PGM CALL**.
- You can call any program by also using the function **Select the cycle** (**SEL CYCLE**).
- As a rule, Q parameters are effective globally with a **PGM CALL**. So please note that changes to Q parameters in the called program also influence the calling program.

Calling a program with PGM CALL

The **PGM CALL** function calls any program as a subprogram. The control runs the called program from the position where it was called in the program.

PGM CALL

- To select the functions for program call, press the **PGM CALL** key

CALL PROGRAM

- Press the **CALL PROGRAM** soft key
- > The control starts the dialog for defining the program to be called.
- Enter the path name with the keyboard

or

SELECT FILE

- Press the **SELECT FILE** soft key
- > The control shows a selection window that allows you to select the program to be called.
- Press the **ENT** key

Calling a program with SEL PGM and CALL SELECTED PGM

Use the function **SEL PGM** to select any program as a subprogram and call it at another position in the program. The control runs the called program from the position where it was called in the program with **CALL SELECTED PGM**.

The **SEL PGM** function is also permitted with string parameters, so that you can dynamically control program calls.

To select the program, proceed as follows:

PGM CALL

- ▶ To select the functions for program call, press the **PGM CALL** key

SELECT PROGRAM

- ▶ Press the **SELECT PROGRAM** soft key
- \> The control starts the dialog for defining the program to be called.

SELECT FILE

- ▶ Press the **SELECT FILE** soft key
- \> The control shows a selection window that allows you to select the program to be called.
- ▶ Press the **ENT** key

To call the selected program, proceed as follows:

PGM CALL

- ▶ To select the functions for program call, press the **PGM CALL** key

CALL SELECTED PROGRAM

- ▶ Press the **CALL SELECTED PROGRAM** soft key
- \> With **CALL SELECTED PGM**, the control calls the last program selected.

> If an NC program that was called using **CALL SELECTED PGM** is missing, then the control interrupts the execution or simulation with an error message. In order to avoid undesired interruptions during program run, all paths to the program beginning can be checked using the **FN 18** function (**ID10 NR110** and **NR111**)
> **Further information:** "FN 18: SYSREAD – Reading system data", page 268

8.5 Nesting

Types of nesting

- Subprogram calls in subprograms
- Program-section repeats within a program-section repeat
- Subprogram calls in program section repeats
- Program-section repeats in subprograms

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: 19
- Maximum nesting depth for main program calls: 19, where a **CYCL CALL** acts like a main program call
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example

0 BEGIN PGM UPGMS MM	
...	
17 CALL LBL "UP1"	Call the subprogram marked with LBL SP1
...	
35 Z+100 R0 FMAX M2	Last program block of the main program with M2
36 LBL "UP1"	Beginning of subprogram SP1
...	
39 CALL LBL 2	Call the subprogram marked with LBL 2
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Beginning of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM SUBPGMS MM	

Program execution

1 Main program UPGMS is executed up to block 17.
2 Subprogram SP1 is called, and executed up to block 39.
3 Subprogram 2 is called, and executed up to block 62. End of subprogram 2 and return jump to the subprogram from which it was called.
4 Subprogram 1 is called, and executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program UPGMS.
5 Main program UPGMS is executed from block 18 up to block 35. Return jump to block 1 and end of program.

Repeating program section repeats

Example

0 BEGIN PGM REPS MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2	Program section call with two repeats
...	
35 CALL LBL 1 REP 1	The program section between this block and LBL 1
...	(block 15) is repeated once
50 END PGM REPS MM	

Program execution

1. Main program REPS is executed up to block 27.
2. Program section between block 27 and block 20 is repeated twice.
3. Main program REPS is executed from block 28 to block 35.
4. Program section between block 35 and block 15 is repeated once (including the program section repeat between 20 and block 27).
5. Main program REPS is executed from block 36 to block 50. Return jump to block 1 and end of program.

Repeating a subprogram

Example

0 BEGIN PGM UPGREP MM	
...	
10 LBL 1	Beginning of program section repeat 1
11 CALL LBL 2	Subprogram call
12 CALL LBL 1 REP 2	Program section call with two repeats
...	
19 Z+100 R0 FMAX M2	Last block of the main program with M2
20 LBL 2	Beginning of subprogram
...	
28 LBL 0	End of subprogram
29 END PGM UPGREP MM	

Program execution

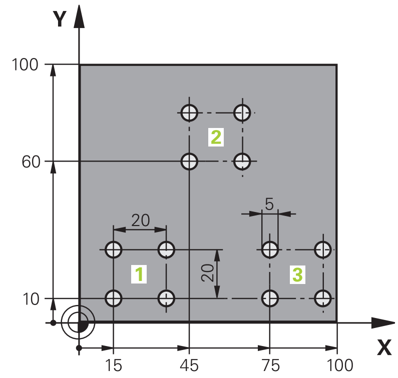
1 Main program UPGREP is executed up to block 11.
2 Subprogram 2 is called and executed.
3 Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.
4 Main program UPGREP is executed from block 13 up to block 19. Return jump to block 1 and end of program.

8.6 Programming examples

Example: Groups of holes

Program run:

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1) in the main program
- Program the group of holes only once in subprogram 1



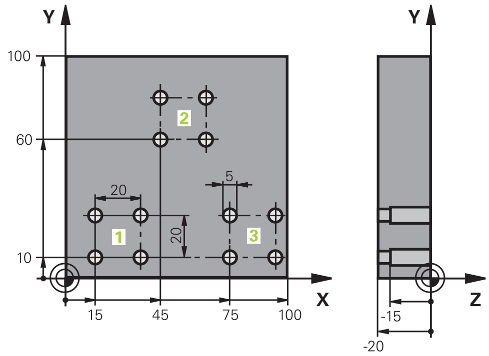
0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3000	Tool call
4 Z+250 R0 FMAX M3	
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q210=+0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DWELL TIME AT DEPTH	
Q395=+0 ;DEPTH REFERENCE	
6 CYCL DEF 7.0 DATUM SHIFT	Datum shift
7 CYCL DEF 7.1 X+15	
8 CYCL DEF 7.2 Y+10	
9 CALL LBL 1	
10 CYCL DEF 7.0 DATUM SHIFT	Datum shift
11 CYCL DEF 7.1 X+75	
12 CYCL DEF 7.2 Y+10	
13 CALL LBL 1	
14 CYCL DEF 7.0 DATUM SHIFT	Datum shift
15 CYCL DEF 7.1 X+45	
16 CYCL DEF 7.2 Y+60	
17 CALL LBL 1	
18 CYCL DEF 7.0 DATUM SHIFT	
19 CYCL DEF 7.1 X+0	

20 CYCL DEF 7.2 Y+0	
21 Z+100 R0 FMAX M30	
22 LBL 1	
23 X+0 R0 FMAX	
24 Y+0 R0 FMAX M99	Move to 1st hole, call cycle
25 X+20 R0 FMAX M99	Move to 2nd hole, call cycle
26 Y+20 R0 FMAX M99	Move to 3rd hole, call cycle
27 X-20 R0 FMAX M99	Move to 4th hole, call cycle
28 LBL 0	
29 END PGM SP2 MM	

Example: Group of holes with several tools

Program run:

- Program the fixed cycles in the main program
- Call the complete hole pattern (subprogram 1) in the main program
- Approach the groups of holes (subprogram 2) in subprogram 1
- Program the group of holes only once in subprogram 2



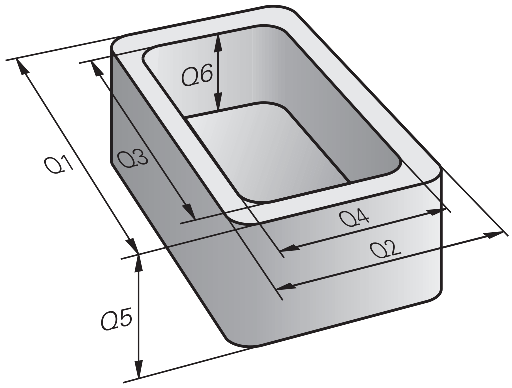
0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Centering drill tool call
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q201=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=3 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 CALL LBL 1	Call subprogram 1 for the entire hole pattern
7 Z+250 R0 FMAX M6	Tool change
8 TOOL CALL 2 Z S4000	Drill tool call
9 FN 0: Q201 = -25	New depth for drilling
10 FN 0: Q202 = +5	New plunging depth for drilling
11 CALL LBL 1	Call subprogram 1 for the entire hole pattern
12 Z+250 R0 FMAX M6	Tool change
13 TOOL CALL 3 Z S500	Reamer tool call

14 CYCL DEF 201 REAMING	Cycle definition: REAMING
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG.	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
15 CALL LBL 1	Call subprogram 1 for the entire hole pattern
16 Z+250 R0 FMAX M2	End of main program
17 LBL 1	Beginning of subprogram 1: Entire hole pattern
18 X+15 R0 FMAX M3	Move to starting point X for hole group 1
19 Y+10 R0 FMAX M3	Move to starting point Y for hole group 1
20 CALL LBL 2	Call subprogram 2 for the hole group
21 X+45 R0 FMAX	Move to starting point X for hole group 2
22 Y+60 R0 FMAX	Move to starting point Y for hole group 2
23 CALL LBL 2	Call subprogram 2 for the hole group
24 X+75 R0 FMAX	Move to starting point X for hole group 3
25 Y+10 R0 FMAX	Move to starting point Y for hole group 3
26 CALL LBL 2	Call subprogram 2 for the hole group
27 LBL 0	End of subprogram 1
28 LBL 2	Beginning of subprogram 2: Group of holes
29 CYCL CALL	1st hole with active fixed cycle
30 IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
31 IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
32 IX-20 R0 FMAX M99	Move to 4th hole, call cycle
33 LBL 0	End of subprogram 2
34 END PGM UP2 MM	

9

Programming Q Parameters

9.1 Principle and overview of functions



With Q parameters you can program entire families of parts in a single NC program by programming variable Q parameters instead of fixed numerical values.

Use Q parameters for e.g.:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

With Q parameters you can also:

- Program contours that are defined through mathematical functions
- Make execution of machining steps depend on certain logical conditions

Q parameters are always identified with letters and numbers. The letters determine the type of Q parameter and the numbers the Q parameter range.

For more information, see the table below:

Q parameter type	Q parameter range	Meaning
Q parameters:		**Parameters affect all NC programs in the control's memory**
	0 – 99	Parameters for the **user**, if there are no overlaps with the HEIDENHAIN-SL cycles
	100 – 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 – 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 – 1399	Parameters that are primarily used with manufacturer cycles when values are given back to the user program
	1400 – 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 – 1999	Parameters for **users**
QL parameters:		**Parameters only effective locally within an NC program**
	0 – 499	Parameters for **users**
QR parameters:		**Parameters permanently (remanence) affect all NC programs in the control's memory, even after a power interruption**
	0 to 99	Parameters for **users**
	100 to 199	Parameters for HEIDENHAIN functions (e.g., cycles)
	200 to 499	Parameters for the machine tool builder (e.g., cycles)

QS parameters (the **S** stands for string) are also available on the control and enable you to process texts.

Q parameter type	Q parameter range	Meaning
QS parameters:		**Parameters affect all NC programs in the control's** memory
	0 – 99	Parameters for the **user**, where no overlaps with the HEIDENHAIN SL cycles are present
	100 – 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 – 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 – 1399	Parameters that are primarily used with manufacturer cycles when values are given back to the user program
	1400 – 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 – 1999	Parameters for **users**

NOTICE

Danger of collision!

Q parameters are used in the HEIDENHAIN cycles, in machine tool builder cycles, and in supplier functions. You can also program Q parameters within the NC program. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.
- Check the machining sequence using a graphic simulation

Programming notes

You can mix Q parameters and numerical values within an NC program.

Q parameters can be assigned numerical values between -999 999 999 and +999 999 999. The input range is limited to 16 digits, of which 9 may be before the decimal point. Internally the control calculates numbers up to a value of 10^{10}.

You can assign a maximum of 255 characters to **QS** parameters.

> The control automatically assigns some Q and QS parameters the same data, e.g., the Q parameter **Q108** is automatically assigned the current tool radius.
>
> **Further information:** " Preassigned Q parameters", page 331
>
> The control saves numerical values internally in a binary number format (standard IEEE 754). Due to the standardized format used, the control does not represent some decimal numbers with a binary number that is 100% exact (round-off error). If you use calculated Q parameter contents for jump commands or positioning moves, then you must take this fact into consideration.

You can reset Q parameters to the status **Undefined**. If a position is programmed with a Q parameter that is undefined, the control ignores this movement.

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 control then displays the following soft keys:

Soft key	Function group	Page
BASIC ARITHM.	Basic arithmetic (assign, add, subtract, multiply, divide, square root)	247
TRIGO- NOMETRY	Trigonometric functions	250
CIRCLE CALCU- LATION	Function for calculating circles	251
JUMP	If/then conditions, jumps	252
DIVERSE FUNCTION	Other functions	256
FORMULA	Entering formulas directly	314

> **i** If you define or assign a Q parameter, then the control shows the **Q**, **QL** and **QR** soft keys. You can use these soft keys to select the desired parameter type. Then you define the parameter number.
>
> If you have a USB keyboard connected, you can press the **Q** key to open the dialog for entering a formula.

9.2 Part families—Q parameters in place of numerical values

Application

The Q parameter function **FN 0: ASSIGN** assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example

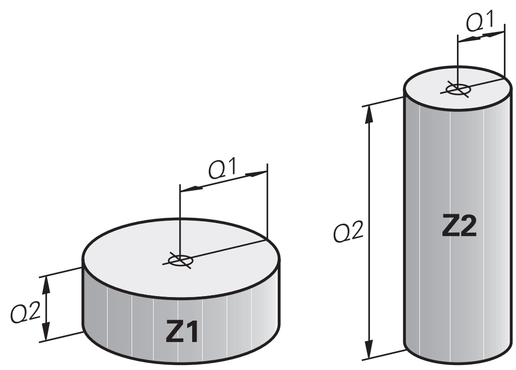
15 FN 0: Q10=25	Assign
...	Q10 is assigned the value 25
25 X +Q10	Means X +25

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



9.3 Describing contours with mathematical functions

Application

The Q parameters listed below enable you to program basic mathematical functions in a machining program:

- Select a Q parameter function: Press the **Q** key (in the numerical keypad on the right). The Q parameter functions are displayed in a soft key row
- To select the basic mathematical functions, press the **BASIC ARITHM...** soft key.
- > The control then displays the following soft keys:

Overview

Soft key	Function
FN0 X = Y	**FN 0**: ASSIGN e. g., **FN 0: Q5 = +60** Directly assign value Reset Q parameter value
FN1 X + Y	**FN 1**: ADDITION e. g., **FN 1: Q1 = -Q2 + -5** Calculate and assign the sum of two values
FN2 X - Y	**FN 2**: SUBTRACTION e. g. **FN 2: Q1 = +10 - +5** Form and assign difference between two values
FN3 X * Y	**FN 3**: MULTIPLICATION e. g. **FN 3: Q2 = +3 * +3** Form and assign the product of two values
FN4 X / Y	**FN 4**: DIVISION e.g., **FN 4: Q4 = +8 DIV +Q2** Calculate and assign the quotient of two values **Not permitted:** Division by 0
FN5 SQRT	**FN 5**: SQUARE ROOT e.g., **FN 5: Q20 = SQRT 4** Calculate and assign the square root of a value **Not permitted:** Square root of a negative value

You can enter the following to the right of the **=** sign:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming fundamental operations

Example 1

Example

16 FN 0: Q5 = +10
17 FN 3: Q12 = +Q5 * +7

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- To select the mathematical functions, press the **BASIC ARITHM.** soft key.

FN0 X = Y

- Select the ASSIGN Q parameter function: Press the **FN0 X = Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **5** (the number of the Q parameter) and confirm with the **ENT** key

FIRST VALUE / PARAMETER?

ENT

- Enter **10**: Assign the numerical value 10 to Q5 and confirm with the **ENT** key

Example 2

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- To select the mathematical functions, press the **BASIC ARITHM.** soft key.

FN3 X * Y

- To select the MULTIPLICATION Q parameter function, press the **FN3 X * Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **12** (the number of the Q parameter) and confirm with the **ENT** key

FIRST VALUE / PARAMETER?

ENT

- Enter **Q5** as the first value and confirm with the **ENT** key.

SECOND VALUE / PARAMETER?

ENT

- Enter **7** as the second value and confirm with the **ENT** key.

Example 3 – Reset Q parameters

Example

16 FN 0: Q5 SET UNDEFINED
17 FN 0: Q1 = Q5

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- To select the mathematical functions, press the **BASIC ARITHM.** soft key.

FN0 X = Y

- Select the ASSIGN Q parameter function: Press the **FN0 X = Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **5** (the number of the Q parameter) and confirm with the **ENT** key

1. VALUE OR PARAMETER?

SET UNDEFINED

- Press **SET UNDEFINED**

> The **FN 0** function also supports transfer of the value **Undefined**. If you wish to transfer the undefined Q parameter without **FN 0**, the control shows the error message **Invalid value**.

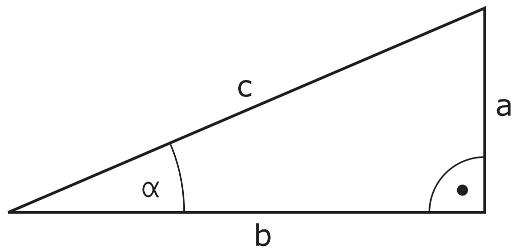
9.4 Angle functions

Definitions

Sine: $\sin \alpha = a / c$

Cosine: $\cos \alpha = b / c$

Tangent: $\tan\alpha = a / b = \sin \alpha / \cos \alpha$



where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

The control can find the angle from the tangent:

$\alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

Example:

a = 25 mm

b = 50 mm

$\alpha = \arctan (a / b) = \arctan 0.5 = 26.57°$

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

Press the **TRIGONOMETRY** soft key to call the trigonometric functions. The control then displays the soft keys listed in the table below:

Soft key	Function
FN6 SIN(X)	**FN 6**: SINUS e. g., **FN 6: Q20 = SIN-Q5** Calculate and assign the sine of an angle in degrees (°)
D7 COS(X)	**FN 7**: COSINE e. g., **FN 7: Q21 = COS-Q5** Calculate and assign the cosine of an angle in degrees (°)
FN8 X LEN Y	**FN 8**: ROOT SUM OF SQUARES e. g., **FN 8: Q10 = +5 LEN +4** Calculate and assign lengths from two values
FN13 X ANG Y	**FN 13**: ANGLE e. g., **FN 13: Q20 = +25 ANG-Q1** Calculate and assign an angle with the arc tangent from the opposite and adjacent sides or with the sine and cosine of the angle (0 < angle < 360°)

9.5 Calculation of circles

Application

The control can use the functions for calculating circles to calculate the circle center and the circle radius from three or four given points on the circle. The calculation is more accurate if four points are used.

Application: These functions can be used, for example, if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Soft key	Function
FN23 3 POINTS OF CIRCLE	FN 23: Determining the CIRCLE DATA from three points e. g., **FN 23: Q20 = CDATA Q30**

The coordinate pairs of three points on a circle must be saved in Q30 and the following five parameters—in this case, up to Q35.

The control then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.

Soft key	Function
FN24 4 POINTS OF CIRCLE	FN 24: Determining the CIRCLE DATA from four points e. g., **FN 24: Q20 = CDATA Q30**

The coordinate pairs of four points on a circle must be saved in Q30 and the following seven parameters—in this case, up to Q37.

The control then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.

> Note that **FN 23** and **FN 24** automatically overwrite the resulting parameter and the two following parameters.

9.6 If-then decisions with Q parameters

Application

The control 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 control continues the program at the label that is programmed after the condition.

Further information: "Labeling subprograms and program section repeats", page 222

If it is not fulfilled, then the control executes the next block.

To call another program as a subprogram, enter a **PGM CALL** program call after the block with the label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

FN 9: IF+10 EQU+10 GOTO LBL1

Abbreviations used:

IF	:	If
EQU	:	Equal to
NE	:	Not equal to
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to
UNDEFINED	:	Undefined
DEFINED	:	Defined

Programming if-then decisions

Possibilities for jump inputs

The following inputs are possible for the condition **IF**:

- Numbers
- Texts
- Q, QL, QR
- **QS** (string parameter)

You have three possibilities for entering the jump address **GOTO**:

- **LBL NAME**
- **LBL NUMBER**
- **QS**

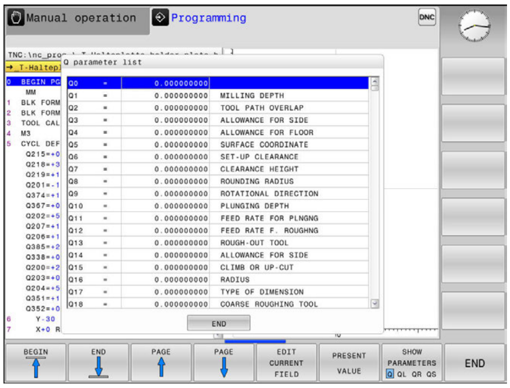
Press the **JUMP** soft key to call the if-then conditions. The control then displays the following soft keys:

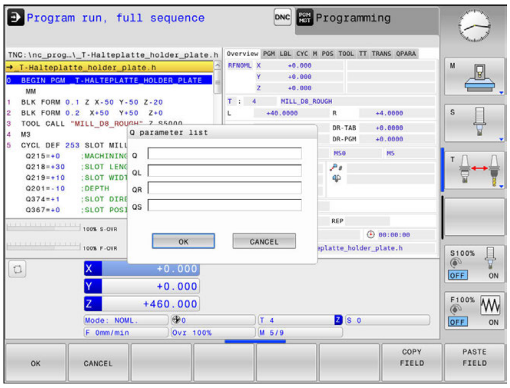
Soft key	Function
FN9 IF X EQ Y GOTO EQU	**FN 9**: IF EQUAL, JUMP e. g. **FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"** If both values or parameters are equal, jump to specified label
FN9 IF X EQ Y GOTO IS UNDEFINED	**FN 9**: IF UNDEFINED, JUMP e. g., **FN 9: IF +Q1 IS UNDEFINED GOTO LBL “UPCAN25“** If the specified parameter is undefined, then a jump is made to the specified label
FN9 IF X EQ Y GOTO IS DEFINED	**FN 9**: IF DEFINED, JUMP e. g., **FN 9: IF +Q1 IS DEFINED GOTO LBL “UPCAN25“** If the specified parameter is defined, then a jump is made to the specified label
FN10 IF X NE Y GOTO	**FN 10**: IF UNEQUAL, JUMP e. g.**FN 10: IF +10 NE -Q5 GOTO LBL 10** If both values or parameters are unequal, jump to specified label
FN11 IF X GT Y GOTO	**FN 11**: IF GREATER, JUMP g. g.**FN 11: IF+Q1 GT+10 GOTO LBL QS5** If the first value or parameter is greater than the second value or parameter, jump to specified label
FN12 IF X LT Y GOTO	**FN 12**: IF LESS, JUMP e. g. **FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME"** If the first value or parameter is smaller than the second value or parameter, jump to specified label

9.7 Checking and changing Q parameters

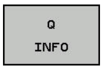
Procedure

You can check Q parameters in all operating modes, and also edit them.





- If you are in a program run, interrupt it if required (e.g., by pressing the **NC STOPP** key and the **INTERNAL STOP** soft key), or stop the test run



- To call the Q parameter functions, press the **Q INFO** soft key or the **Q** key
- > The control lists all of the parameters and their corresponding current values.
- Use the arrow keys or the **GOTO** key to select the desired parameter.
- If you would like to change the value, press the **EDIT CURRENT FIELD** soft key. Enter a new file name and confirm with **ENT**
- To leave the value unchanged, press the **PRESENT VALUE** soft key or end the dialog with the **END** key

ℹ	All of the parameters with displayed comments are used by the control within cycles or as transfer parameters. If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The control then displays the specific parameter type. The functions previously described also apply.

You can have Q parameters also displayed in the additional status display in all operating modes (except **Programming** mode).

- If you are in a program run, interrupt it if required (e.g. by pressing the**NC-STOPP** key and the **INTERNAL STOP** soft key), or stop the test run
- Call the soft key row for screen layout

PROGRAM + STATUS

- Select the layout option for the additional status display
- > In the right half of the screen, the control shows the **Overview** status form.

STATUS OF Q PARAM.

- Press the **STATUS OF Q PARAM.** soft key

Q PARAMETER LIST

- Press the **Q PARAMETER LIST** soft key.
- > The control opens a pop-up window.
- For each parameter type (Q, QL, QR, QS), define the parameter numbers you wish to control. Separate single Q parameters with a comma, and connect sequential Q parameters with a hyphen, e.g. 1,3,200-208. The input range per parameter type is 132 characters

> The display in the **QPARA** tab always contains eight decimal places. The result of Q1 = COS 89.999 is shown by the control as 0.00001745, for example. Very large or very small values are displayed by the control in exponential notation. The result of Q1 = COS 89.999 * 0.001 is shown by the control as +1.74532925e-08, whereby e-08 corresponds to the factor of 10^{-8}.

9.8 Additional functions

Overview

Press the **DIVERSE FUNCTION** soft key to call the additional functions. The control then displays the following soft keys:

Soft key	Function	Page
FN14 ERROR=	**FN 14: ERROR** Display error messages	257
FN16 F-PRINT	**FN 16: F-PRINT** Formatted output of texts or Q parameter values	261
FN18 SYS-DATUM READ	**FN 18: SYSREAD** Read system data	268
FN19 PLC=	**FN 19: PLC** Transfer values to the PLC	297
FN20 WAIT FOR	**FN 20: WAIT FOR** NC and PLC synchronization	298
FN26 OPEN TABLE	**FN 26: TABOPEN** Open a freely definable table	354
FN27 WRITE TO TABLE	**FN 27: TABWRITE** Write to a freely definable table	354
FN28 READ FROM TABLE	**FN 28: TABREAD** Read from a freely definable table	355
FN29 PLC LIST=	**FN 29: PLC** Transfer up to eight values to the PLC	298
FN37 EXPORT	**FN 37: EXPORT**Export local Q parameters or QS parameters into a calling program	299
FN38 SEND	**FN 38: SEND** Send information from the NC program	299

FN 14: ERROR: Displaying error messages

With the **FN 14: ERROR** error function, you can output error messages under program control. The messages are predefined by the machine tool builder or by HEIDENHAIN. If, during a program run or test run, the control encounters a block with **FN 14: ERROR**, then the control will interrupt the program run or test run and display an error message. The program must then be restarted.

Error numbers area	Standard dialog
0 ... 999	Machine-dependent dialog
1000 ... 1199	Internal error messages

Example

The control is intended to display a message if the spindle is not switched on.

```
180 FN 14: ERROR = 1000
```

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 FACTOR not permitted
1008	MIRROR IMAGE not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Input value incorrect
1012	Incorrect sign
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory input
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined

Error number	Text
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive nesting
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	Q218 must be greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted
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

Error number	Text
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 to 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter Q351 unequal to 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as negative
1078	Q303 in meas. cycle undefined!
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory meas. points
1082	Incorrect clearance height
1083	Contradictory plunge type
1084	This fixed cycle not allowed
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not allowed
1090	Enter an infeed not equal to 0
1091	Switchover of Q399 not allowed
1092	Tool not defined
1093	Tool number not permitted

Error number	Text
1094	Tool name not permitted
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed
1100	Kinematic access not possible
1101	Meas. pos. not in traverse range
1102	Preset compensation not possible
1103	Tool radius too large
1104	Plunging type is not possible
1105	Plunge angle incorrectly defined
1106	Angular length is undefined
1107	Slot width is too large
1108	Scaling factors not equal
1109	Tool data inconsistent

FN16: F-PRINT – Formatted output of texts and Q parameter values

With **FN 16: F-PRINT**, you can output any messages from your NC program on the screen. The control displays such messages in a pop-up window.

Further information: "Displaying messages on the control's screen", page 266

With the function **FN 16: F-PRINT**, you can save Q parameter values and output formatted texts (e.g. in order to save measurement reports). If you output the values, then the control saves the data in the file that you define in the **FN 16** block. The maximum size of the output file is 20 kB.

To be able to use the function **FN 16: F-PRINT**, first program a text file that specifies the output format.

Available functions

Use the following formatting functions for creating a text file:

Special characters	Function
"..........."	Define output format for texts and variables between the quotation marks
%9.3F	Format for Q parameter: ■ Define %: format ■ 9.3: Total of 9 characters (incl. decimal point), of which 3 are decimal places ■ F: Floating (decimal number), format for Q, QL, QR
%+7.3F	Format for Q parameter: ■ Define %: format ■ +: number right-aligned ■ 7.3: Total of 7 characters (incl. decimal point), of which 3 are decimal places ■ F: Floating (decimal number), format for Q, QL, QR
%S	Format for text variable QS
%D or **%I**	Format for integer
,	Separation character between output format and parameter
;	End of block character
\n	Line break
+	Q parameter value, right-aligned
-	Q parameter value, left-aligned

The following functions allow you to include the following additional information in the protocol log file:

Keyword	Function
CALL_PATH	Indicates the path for the NC program where you will find the FN16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with FN16. Example: M_CLOSE;
M_APPEND	Upon renewed output, appends the log to the existing log. Example: M_APPEND;
L_ENGLISH	Outputs text only for English conversational language
L_GERMAN	Outputs text only for German conversational language
L_CZECH	Outputs text only for Czech conversational language
L_FRENCH	Outputs text only for French conversational language
L_ITALIAN	Outputs text only for Italian conversational language
L_SPANISH	Outputs text only for Spanish conversational language
L_PORTUGUE	Outputs text only for Portuguese conversational language
L_SWEDISH	Outputs text only for Swedish conversational language
L_DANISH	Outputs text only for Danish conversational language
L_FINNISH	Outputs text only for Finnish conversational language
L_DUTCH	Outputs text only for Dutch conversational language
L_POLISH	Outputs text only for Polish conversational language
L_HUNGARIA	Outputs text only for Hungarian conversational language
L_CHINESE	Outputs text only for Chinese conversational language
L_CHINESE_TRAD	Outputs text only for Chinese (traditional) conversational language
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_NORWEGIAN	Outputs text only for Norwegian conversational language
L_ROMANIAN	Outputs text only for Romanian conversational language

Keyword	Function
L_SLOVAK	Outputs text only for Slovakian conversational language
L_TURKISH	Outputs text only for Turkish conversational language
L_ALL	Display text independently of the conversational language
HOUR	Number of hours from the real-time clock
MIN	Number of minutes from the real-time clock
SEC	Number of seconds from the real-time clock
DAY	Day from the real-time clock
MONTH	Month as a number from the real-time clock
STR_MONTH	Month as a string abbreviation from the real-time clock
YEAR2	Two-digit year from the real-time clock
YEAR4	Four-digit year from the real-time clock

Creating a text file

To output the formatted texts and Q-parameter values, create a text file with the control's text editor. In this file you then define the output format and Q parameters you want to output. Create this file with the extension **.A**.

Example of a text file to define the output format:

```
"MEASURING LOG OF IMPELLER CENTER OF GRAVITY";
"DATE: %02d.%02d.%04d",DAY,MONTH,YEAR4;
"TIME: %02d:%02d:%02d",HOUR,MIN,SEC;
"NO. OF MEASURED VALUES: = 1";
"X1 = %9.3F", Q31;
"Y1 = %9.3F", Q32;
"Z1 = %9.3F", Q33;
```

In the NC program, program FN 16: F-PRINT to activate the output:

Enter the path of the source and the path of the output file in the FN 16 function .

Specify the output file containing the output texts within the function **FN16**. The control generates the output file at the end of program (**END PGM**), at program abortion (**NC-STOPP** key) or via **M_CLOSE** command.

> If you only specify the file name as the path name of the log file, then the control saves the log file in the directory of the NC program with the **FN16** function.
>
> Program relative paths as an alternative to complete paths:
>
> - Starting from the folder of the calling file one folder level down **FN 16: F-PRINT MASKE\MASKE1.A/ PROT\PROT1.TXT**
> - Starting from the folder of calling file one folder level up and in another folder **FN 16: F-PRINT ..\MASKE \MASKE1.A/ ..\PROT1.TXT**

Example

```
96 FN 16: F-PRINT TNC:\MASK\MASK1.A/ TNC:\PROT1.TXT
```

The control then creates the file PROT1.TXT:

MEASURING LOG OF IMPELLER CENTER OF GRAVITY

DATE: July 15, 2015

TIME: 8:56:34 AM

NO. OF MEASURED VALUES : = 1

X1 = 149.360

Y1 = 25.509

Z1 = 37.000

Operating and programming notes:

- If you output the same file multiple times in the program, then, within the target file, the control adds the current output after the previously output contents.
- In the **FN16** block, program the format file and the log file with their respective file type extensions.
- The file name extension of the log file determines the file format of the output (e.g., TXT, .A, .XLS, .HTML).
- In machine parameters **fn16DefaultPath** (no. 102202) and **fn16DefaultPathSim** (no. 102203) you can define a default path for outputting log files.
- If you use **FN16** the file must not be UTF8-encoded.
- You receive a great deal of relevant and interesting information for a log file by means of the function **FN 18** (e.g., the number of the last touch probe cycle used).
  Further information: "FN 18: SYSREAD – Reading system data", page 268

Displaying messages on the control's screen

You can also use the function **FN16: F-PRINT** to display any messages from the NC program in a pop-up window on the control's screen. This makes it easy to display explanatory texts, including long texts, at any point in the program in a way that the user has to react to them. You can also display Q-parameter contents if the protocol description file contains such instructions.

For the message to appear on the control's screen, you need only enter **screen:** as the name of the protocol file.

Example

```
96 FN 16: F-PRINT TNC:\MASK\MASK1.A/screen:
```

If the message has more lines than fit in the pop-up window, you can use the arrow keys to page in the window.

To close the pop-up window, press the **CE** key. To have the program close the window, program the following NC block:

Example

```
96 FN 16: F-PRINT TNC:\MASK\MASK1.A/SCLR:
```

> The **FN16** function overwrites log files that are present by default and that have the same name. If, during renewed output, you would like to add logs to existing logs, then use **M_APPEND**.

Exporting messages

The **FN 16** function also enables you to save the log files externally.

Enter the complete target path in the **FN 16** function:

Example

```
96 FN 16: F-PRINT TNC:\MSK\MSK1.A / PC325:\LOG\PRO1.TXT
```

> The **FN16** function overwrites log files that are present by default and that have the same name. If, during renewed output, you would like to add logs to existing logs, then use **M_APPEND**.

Entering the source or the target with parameters

You can enter the source file and the output file as Q parameters or as QS parameters. For this purpose you previously define the desired parameter in the NC program.

Further information: "Assign string parameters", page 319

In order for the control to recognize that you are working with Q parameters, enter them in the **FN16-**function with the following syntax:

Input	Function
:'QS1'	Set the QS parameter with preceding colon and between single quotation marks
:'QL3'.txt	Specify additional file name extension for the target file if required

Printing messages

You can also use the function **FN16: F-PRINT** to print any messages on a connected printer.

Further information: "Printer", page 97

In order for the messages to be sent to the printer, you must enter **Printer:** as the name of the log file and then enter the corresponding file name.

The control saves the file in the **PRINTER:** path until the file is printed.

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/PRINTER:\DRUCK1
```

FN 18: SYSREAD – Reading system data

With the **FN 18: SYSREAD** function you can read system data and store them in Q parameters. The selection of the system datum occurs via a group number (ID no.), a system data number, and, if necessary, an index.

The read values of the function **FN 18: SYSREAD** are always output by the control in **metric** units regardless of the NC program's unit of measure.

The following is a complete list of the **FN 18: SYSREAD** function. Please be aware that not all functions are available depending on the model of your control.

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Program information				
	10	3	-	Number of the active machining cycle
		6	-	Number of the most recently executed touch probe cycle –1 = None
		7	-	Type of calling NC program: –1 = None 0 = Visible NC program 1 = Cycle/macro, main program is visible 2 = Cycle/macro, there is no visible main program
		103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
		110	QS parameter number	Is there a file with the name QS(IDX)? 0 = No, 1 = Yes This function eliminates relative file paths.
		111	QS parameter number	Is there a directory with the name QS(IDX)? 0 = no, 1 = Yes Only absolute directory paths are possible.

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Branch addresses of the system				
	13	1	-	Label jumped to during M2/M30 instead of ending the current program. Value = 0: M2/M30 have the normal effect
		2	-	Label jumped to in the event of FN14: ERROR with the NC CANCEL reaction instead of aborting the program with an error message. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
		3	-	Label jumped to in the event of an internal server error (SQL, PLC, CFG) or with erroneous file operations (FUNCTION FILECOPY, FUNCTION FILEMOVE, or FUNCTION FILEDELETE) instead of aborting the program with an error message. Value = 0: Error has the normal effect.
Machine status				
	20	1	-	Active tool number
		2	-	Prepared tool number
		3	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W
		4	-	Programmed spindle speed
		5	-	Active spindle condition -1 = spindle condition not defined 0 = M3 active 1 = M4 active 2 = M5 active after M3 3 = M5 active after M4
		7	-	Active gear range
		8	-	Active coolant status 0 = off, 1 = on
		9	-	Active feed rate
		10	-	Index of prepared tool
		11	-	Index of active tool
		14	-	Number of active spindle
		20	-	Programmed cutting speed in turning operation
		21	-	Spindle mode in turning mode: 0 = constant speed 1 = constant cutting speed
		22	-	Coolant status M7: 0 = inactive, 1 = active

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		23	-	Coolant status M8: 0 = inactive, 1 = active
Channel data				
	25	1	-	Channel number
Cycle parameters				
	30	1	-	Set-up clearance
		2	-	Hole depth / milling depth
		3	-	Plunging depth
		4	-	Feed rate for plunging
		5	-	First side length of pocket
		6	-	Second side length of pocket
		7	-	First side length of slot
		8	-	Second side length of slot
		9	-	Radius of circular pocket
		10	-	Feed rate for milling
		11	-	Rotational direction of the milling path
		12	-	Dwell time
		13	-	Thread pitch for Cycles 17 and 18
		14	-	Finishing allowance
		15	-	Roughing angle
		21	-	Probing angle
		22	-	Probing path
		23	-	Probing feed rate
		49	-	HSC mode (Cycle 32 Tolerance)
		50	-	Tolerance for rotary axes (Cycle 32 Tolerance)
		52	Q parameter number	Type of transfer parameter for user cycles: –1: Cycle parameter not programmed in CYCL DEF 0: Cycle parameter numerically programmed in CYCL DEF (Q parameter) 1: Cycle parameter programmed as string in CYCL DEF (Q parameter)
		60	-	Clearance height (touch probe cycles 30 to 33)
		61	-	Inspection (touch probe cycles 30 to 33)
		62	-	Cutting edge measurement (touch probe cycles 30 to 33)
		63	-	Q parameter number for the result (touch probe cycles 30 to 33)
		64	-	Q parameter type for the result (touch probe cycles 30 to 33) 1 = Q, 2 = QL, 3 = QR

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		70	-	Multiplier for feed rate (cycles 17 and 18)
Modal status				
	35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
Data for SQL tables				
	40	1	-	Result code for the last SQL command. If the last result code was 1 (=error), the error code is transferred as the return code.
Data from the tool table				
	50	1	Tool no.	Tool length L
		2	Tool no.	Tool radius R
		3	Tool no.	Tool radius R2
		4	Tool no.	Oversize for tool length DL
		5	Tool no.	Tool radius oversize DR
		6	Tool no.	Tool radius oversize DR2
		7	Tool no.	Tool locked TL 0 = not locked, 1 = locked
		8	Tool no.	Number of the replacement tool RT
		9	Tool no.	Maximum tool age TIME1
		10	Tool no.	Maximum tool age TIME2
		11	Tool no.	Current tool age CUR.TIME
		12	Tool no.	PLC status
		13	Tool no.	Maximum tooth length LCUTS
		14	Tool no.	Maximum plunge angle ANGLE
		15	Tool no.	TT: Number of tool teeth CUT
		16	Tool no.	TT: Wear tolerance for length, LTOL
		17	Tool no.	TT: Wear tolerance for radius, RTOL
		18	Tool no.	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	Tool no.	TT: Offset in plane R-OFFS R = 99999.9999
		20	Tool no.	TT: Offset in length L-OFFS
		21	Tool no.	TT: Breakage tolerance for length, LBREAK
		22	Tool no.	TT: Breakage tolerance for radius, RBREAK
		28	Tool no.	Maximum speed NMAX
		32	Tool no.	Point angle TANGLE
		34	Tool no.	LIFTOFF allowed (0 = No, 1 = Yes)
		35	Tool no.	Wear tolerance for radius R2TOL

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		36	Tool no.	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	Tool no.	Corresponding line in the touch-probe table
		38	Tool no.	Timestamp of last use
		40	Tool no.	Pitch for thread cycles
Data from the pocket table				
	51	1	Pocket number	Tool number
		2	Pocket number	0 = no special tool 1 = special tool
		3	Pocket number	0 = no fixed pocket 1 = fixed pocket
		4	Pocket number	0 = pocket not locked 1 = pocket locked
		5	Pocket number	PLC status
Determine the tool pocket				
	52	1	Tool no.	Pocket number
		2	Tool no.	Tool magazine number
Tool data for T and S strobes				
	57	1	T code	Tool number IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		2	T code	Tool index IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		5	-	Spindle speed IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
Values programmed in TOOL CALL				
	60	1	-	Tool number T
		2	-	Active tool axis 0 = X 1 = Y 2 = Z 6 = U 7 = V 8 = W
		3	-	Spindle speed S
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Automatic TOOL CALL 0 = Yes, 1 = No
		7	-	Tool radius oversize DR2

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		8	-	Tool index
		9	-	Active feed rate
		10	-	Cutting speed [mm/min]
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Values for LAC and VSC				
	71	0	2	Total inertia determined by the LAC weighing run in [kgm²] (with A/B/C rotary axes) or total mass in [kg] (with X/Y/Z linear axes)
		1	0	Cycle 957 Retraction from thread
Freely available memory area for OEM cycles				
	72	0-39	0 to 30	Freely available memory area for OEM cycles. The values are only reset by the control during a control reboot (= 0). With "Cancel," the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Freely available memory area for user cycles				
	73	0-39	0 to 30	Freely available memory area for user cycles The values are only reset by the control during a control reboot (= 0). With "Cancel," the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Minimum spindle speed				
	90	1	Spindle ID	Minimum spindle speed of the lowest gear range. If no gear ranges are configured, the spindle speed is taken from the parameter set with index 0. Index 99 = active spindle
Tool compensation				
	200	1	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active radius
		2	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active length

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
		6	Tool no.	Tool length Index 0= active tool
Coordinate transformations				
	210	1	-	Basic rotation (manual)
		2	-	Programmed rotation
		3	-	Active mirror axis. Bits 0 to 2 and 6 to 8: Axes X, Y, Z and U, V, W
		4	Axis	Active scaling factor Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 - 3 (A, B, C)
		6	-	Tilt working plane in Program Run operating modes 0 = Not active –1 = Active
		7	-	Tilt working plane in Manual operating modes 0 = Not active –1 = Active
		8	QL parameter no.	Angle of misalignment between spindle and tilted coordinate system. Projects the angle specified in the QL parameter from the input coordinate system to the tool coordinate system. If IDX is omitted, the angle 0 is used for projection.

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Active coordinate system				
	211	–	-	1 = input system (default) 2 = REF system 3 = tool change system
Special transformations in turning mode				
	215	1	-	Angle for the precession of the input system in the XY plane in turning mode To reset the transformation the value 0 must be entered for the angle. This transformation is used in connection with Cycle 800 (parameter Q497).
		3	1-3	Reading out of the spatial angle written with NR2 Index: 1 - 3 (redA, redB, redC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read values for OEM offset. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	-	Software limit switch on or off: 0 = on, 1 = off For modulo axes, either both the upper and lower limits or no limit at all must be set.
		12	Axis	Persistently overwrite the value for the negative software limit switch in CfgPosition-Limits. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		13	Axis	Persistently overwrite the value for the positive software limit switch in CfgPosition-Limits. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
Read the nominal position in the REF system				
	240	1	Axis	Current nominal position in the REF system
Read the nominal position in the REF system, including offsets (handwheel, etc.)				
	241	1	Axis	Current nominal position in the REF system
Read the current position in the active coordinate system				
	270	1	Axis	Current nominal position in the input system
Read the current position in the active coordinate system, including offsets (handwheel, etc.)				

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
	271	1	Axis	Current nominal position in the input system
Read information to M128				
	280	1	-	M128 active: –1 = Yes, 0 = No
Machine kinematics				
	290	5	-	0: Temperature compensation not active 1: Temperature compensation active
		10	-	Index of the machine kinematics from Channels/ChannelSettings/CfgKin-List/kinCompositeModels programmed in FUNCTION MODE MILL or FUNCTION MODE TURN –1 = Not programmed.
Read data of the machine kinematics				
	295	1	QS parameter no.	Read the axis names of the active 3-axis kinematics. The axis names are written according to QS(IDX), QS(IDX+1), and QS(IDX+2). 0 = Operation successful
		2	0	Is FACING HEAD POS function active? 1 = Yes, 0 = No
		4	Rotary axis	Read whether the defined rotary axis partici-pates in the kinematic calculation. 1 = Yes, 0 = No (A rotary axis can be excluded from the kinematics calculating using M138.) Index: 4, 5, 6 (A, B, C)
		10	Axis	Determine programmable axes. Determine the axis ID associated with the specified axis index (index from CfgAxis/axisList). Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		11	Axis ID	Determine programmable axes. Determine the index of the axis (X = 1, Y = 2, ...) for the specified axis ID Index: Axis ID (index from CfgAxis/axisList)
Modify the geometrical behavior				
	310	20	Axis	Diameter programming: –1 = on, 0 = off
Current system time				
	320	1	0	System time in seconds that has elapsed since 01.01.1970, 00:00:00 (real time).
			1	System time in seconds that has elapsed since 01.01.1970, 00:00:00 (look-ahead calcu-lation).
		3	-	Read the processing time of the current NC program.

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Formatting of system time				
	321	0	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm:ss
		1	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm:ss
		2	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm
		3	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY h:mm
		4	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm:ss
		5	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		6	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD h:mm
		7	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD h:mm
		8	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY
		9	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY
		10	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY
		11	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		12	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD
		13	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: hh:mm:ss
		14	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm:ss
		15	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Global Program Settings (GPS): Global activation status				
	330	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
		1	-	GPS: Basic rotation 0 = Off, 1 = On
		3	Axis	GPS: Mirroring 0 = Off, 1 = On Index: 1 - 6 (X, Y, Z, A, B, C)
		4	-	GPS: Shift in the modified workpiece system 0 = Off, 1 = On
		5	-	GPS: Rotation in input system 0 = Off, 1 = On
		6	-	GPS: Feed rate factor 0 = Off, 1 = On
		8	-	GPS: Handwheel superimpositioning 0 = Off, 1 = On
		10	-	GPS: Virtual tool axis VT 0 = Off, 1 = On
		15	-	GPS: Selection of the handwheel coordinate system 0 = Machine coordinate system M-CS 1 = Workpiece coordinate system W-CS 2 = Modified workpiece coordinate system mW-CS 3 = Working plane coordinate system WPL-CS
		16	-	GPS: Shift in the workpiece system 0 = Off, 1 = On
		17	-	GPS: Axis offset 0 = Off, 1 = On

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 - 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 - 6 (X, Y, Z, A, B, C)
		5	-	GPS: Angle of rotation in input coordinate system I-CS
		6	-	GPS: Feed rate factor
		8	Axis	GPS: Handwheel superimpositioning Maximum value Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		9	Axis	GPS: Value for handwheel superimpositioning Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 - 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 - 6 (A, B, C)
TS touch trigger probe				
	350	50	1	Touch probe type: 0: TS120, 1: TS220, 2: TS440, 3: TS630, 4: TS632, 5: TS640, 6: TS444, 7: TS740
			2	Line in the touch-probe table
		51	-	Effective length
		52	1	Effective radius of the stylus tip
			2	Rounding radius
		53	1	Center offset (reference axis)
			2	Center offset (minor axis)
		54	-	Spindle-orientation angle in degrees (center offset)
		55	1	Rapid traverse
			2	Measuring feed rate
			3	Feed rate for pre-positioning: FMAX_PROBE or FMAX_MACHINE
		56	1	Maximum measuring range
			2	Set-up clearance
		57	1	Spindle orientation possible 0=No, 1=Yes
			2	Angle of spindle orientation in degrees

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
		71	1/2/3	TT: Touch probe center (REF system)
		72	-	TT: Touch probe radius
		75	1	TT: Rapid traverse
			2	TT: Measuring feed rate with stationary spindle
			3	TT: Measuring feed rate with rotating spindle
		76	1	TT: Maximum probing path
			2	TT: Safety clearance for linear measurement
			3	TT: Safety clearance for radius measurement
			4	TT: Distance from the lower edge of the cutter to the upper edge of the stylus
		77	-	TT: Spindle speed
		78	-	TT: Probing direction
		79	-	TT: Activate radio transmission
		80	-	TT: Stop probing movement upon stylus deflection
Preset from touch probe cycle (probing results)				
	360	1	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (input coordinate system). Compensations: length, radius, and center offset
		2	Axis	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (machine coordinate system, only axes from the active 3-D kinematics are allowed as index). Compensation: only center offset
		3	Coordinate	Result of measurement in the input system of touch probe Cycles 0 and 1. The measurement result is read out in the form of coordinates. Compensation: only center offset
		4	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (workpiece coordinate system) The measurement result is read in the form of coordinates. Compensation: only center offset
		5	Axis	Axis values, not compensated
		6	Coordinate / axis	Readout of the measurement results in the form of coordinates / axis values in the input system from probing operations. Compensation: only length
		10	-	Oriented spindle stop

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		11	-	Error status of probing: 0: Probing was successful –1: Touch point not reached –2: Touch probe already deflected at the start of the probing process
Read values from or write values to the active datum table				
	500	Row number	Column	Read values
Read values from or write values to the preset table (basic transformation)				
	507	Row number	1-6	Read values
Read axis offsets from or write axis offsets to the preset table				
	508	Row number	1-9	Read values
Data for pallet machining				
	510	1	-	Active line
		2	-	Pallet number from the PAL/PGM field
		3	-	Active row of the pallet table.
		4	-	Last line of the NC program for the current pallet.
		5	Axis	Tool-oriented editing: Clearance height is programmed: 0 = No, 1 = Yes Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		6	Axis	Tool-oriented editing: Clearance height The value is invalid if ID510 NR5 returns the value 0 with the corresponding IDX. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		10	-	Row number up to which the pallet table is to be searched during block scan.
		20	-	Type of pallet editing? 0 = Workpiece-oriented 1 = Tool oriented
		21	-	Automatic continuation after NC error: 0 = Locked 1 = Active 10 = Abort continuation 11 = Continuation with the rows in the pallet table that would have been executed next if not for the NC error 12 = Continuation with the row in the pallet table in which the NC error arose 13 = Continuation with the next pallet

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Read data from the point table				
	520	Row number	1-3 X/Y/Z	Read value from active point table.
			10	Read value from active point table.
			11	Read value from active point table.
Read or write the active preset				
	530	1	-	Number of the active preset in the active preset table.
Active pallet preset				
	540	1	-	Number of the active pallet preset. Returns the number of the active preset. If no pallet preset is active, the function returns the value –1.
		2	-	Number of the active pallet preset. As with NR1.
Values for the basic transformation of the pallet preset				
	547	row number	Axis	Read values of the basic transformation from the pallet preset table. Index: 1 - 6 (X, Y, Z, SPA, SPB, SPC)
Axis offsets from the pallet preset table				
	548	Row number	Offset	Read values of the axis offsets from the pallet preset table. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Read and write the machine status				
	590	2	1-30	Freely available; not deleted during program selection.
		3	1-30	Freely available; not deleted during a power failure (persistent storage).
Read/write look-ahead parameter of a single axis (at machine level)				
	610	1	-	Minimum feed rate (**MP_minPathFeed**) in mm/min
		2	-	Minimum feed rate at corners (**MP_minCornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPathJerk**) in m/s^3
		5	-	Max. jerk at high speeds (**MP_maxPathJerkHi**) in m/s^3
		6	-	Tolerance at low speeds (**MP_pathTolerance**) in mm

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		7	-	Tolerance at high speeds (**MP_pathToleranceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPathYank**) in m/s^4
		9	-	Tolerance factor for curve machining (**MP_curveTolFactor**)
		10	-	Factor for max. permissible jerk at curvature changes (**MP_curveJerkFactor**)
		11	-	Maximum jerk with probing movements (**MP_pathMeasJerk**)
		12	-	Angle tolerance for machining feed rate (**MP_angleTolerance**)
		13	-	Angle tolerance for rapid traverse (**MP_angleToleranceHi**)
		14	-	Max. corner angle for polygons (**MP_maxPolyAngle**)
		18	-	Radial acceleration with machining feed rate (**MP_maxTransAcc**)
		19	-	Radial acceleration with rapid traverse (**MP_maxTransAccHi**)
		20	Index of physical axis	Max. feed rate (**MP_maxFeed**) in mm/min
		21	Index of physical axis	Max. acceleration (**MP_maxAcceleration**) in m/s^2
		22	Index of physical axis	Maximum transition jerk of the axis in rapid traverse (**MP_axTransJerkHi**) in m/s^2
		23	Index of physical axis	Maximum transition jerk of the axis during machining free rate (**MP_axTransJerk**) in m/s^3
		24	Index of physical axis	Acceleration feedforward control (**MP_compAcc**)
		25	Index of physical axis	Axis-specific jerk at low speeds (**MP_axPathJerk**) in m/s^3
		26	Index of physical axis	Axis-specific jerk at high speeds (**MP_axPathJerkHi**) in m/s^3
		27	Index of physical axis	More precise tolerance examination in corners (**MP_reduceCornerFeed**) 0 = deactivated, 1 = activated
		28	Index of physical axis	DCM: Maximum tolerance for linear axes in mm (**MP_maxLinearTolerance**)
		29	Index of physical axis	DCM: Maximum angle tolerance in [°] (**MP_maxAngleTolerance**)
		30	Index of physical axis	Tolerance monitoring for successive threads (**MP_threadTolerance**)

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		31	Index of physical axis	Form (**MP_shape**) of the **axisCutterLoc** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		32	Index of physical axis	Frequency **MP_frequency**) of the **axisCutterLoc** filter in Hz
		33	Index of physical axis	Form (**MP_shape**) of the **axisPosition** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		34	Index of physical axis	Frequency (**MP_frequency**) of the **axisPosition** filter in Hz
		35	Index of physical axis	Order of the filter for **Manual** operating mode (**MP_manualFilterOrder**)
		36	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisCutterLoc** filter
		37	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisPosition** filter
		38	Index of physical axis	Axis-specific jerk for probing movements (**MP_axMeasJerk**)
		39	Index of physical axis	Weighting of the filter error for calculating filter deviation (**MP_axFilterErrWeight**)
		40	Index of physical axis	Maximum filter length of position filter (**MP_maxHscOrder**)
		41	Index of physical axis	Maximum filter length of CLP filter (**MP_maxHscOrder**)
		42	-	Maximum feed rate of the axis at machining feed rate (**MP_maxWorkFeed**)
		43	-	Maximum path acceleration at machining feed rate (**MP_maxPathAcc**)
		44	-	Maximum path acceleration at rapid traverse (**MP_maxPathAccHi**)
		51	Index of physical axis	Compensation of following error in the jerk phase (**MP_IpcJerkFact**)
		52	Index of physical axis	kv factor of the position controller in 1/s (**MP_kvFactor**)

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Measure the maximum utilization of an axis				
	621	0	Index of physical axis	Conclude measurement of the dynamic load and save the result in the specified Q parameter.
Read SIK contents				
	630	0	Option no.	You can explicitly determine whether the SIK option given under **IDX** has been set or not. 1 = option is enabled 0 = option is not enabled
		1	-	You can determine whether a Feature Content Level (for upgrade functions) is set, and which one. –1 = No FCL is set <No.> = FCL that is set
		2	-	Read serial number of the SIK -1 = No valid SIK in the system
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)
Workpiece counter				
	920	1	-	Planned workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		2	-	Already machined workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		12	-	Workpieces still to be machined. In **Test Run** operating mode the counter generally generates the value 0.
Read and write data of current tool				
	950	1	-	Tool length L
		2	-	Tool radius R
		3	-	Tool radius R2
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Tool radius oversize DR2
		7	-	Tool locked TL 0 = not locked, 1 = locked
		8	-	Number of the replacement tool RT
		9	-	Maximum tool age TIME1
		10	-	Maximum tool age TIME2 at TOOL CALL
		11	-	Current tool age CUR.TIME
		12	-	PLC status

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		13	-	Tooth length in the tool axis LCUTS
		14	-	Maximum plunge angle ANGLE
		15	-	TT: Number of tool teeth CUT
		16	-	TT: Wear tolerance for length LTOL
		17	-	TT: Wear tolerance for radius RTOL
		18	-	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	-	TT: Offset in plane R-OFFS R = 99999.9999
		20	-	TT: Offset in length L-OFFS
		21	-	TT: Break tolerance for length LBREAK
		22	-	TT: Break tolerance for radius RBREAK
		28	-	Maximum spindle speed [rpm] NMAX
		32	-	Point angle TANGLE
		34	-	LIFTOFF allowed (0 = No, 1 = Yes)
		35	-	Wear tolerance for radius R2TOL
		36	-	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	-	Corresponding line in the touch-probe table
		38	-	Timestamp of last use
		39	-	ACC
		40	-	Pitch for thread cycles
		44	-	Exceeding the tool life
Freely available memory area for tool management				
	956	0-9	-	Freely available data area for tool management. The data is not reset when the program is aborted.
Transformation data for general tools				
	960	1	-	Position within the tool system explicitly defined:
		2	-	Position defined by directions:
		3	-	Shift in X
		4	-	Shift in Y
		5	-	Shift in Z
		6	-	X component of the Z direction
		7	-	Y component of the Z direction
		8	-	Z component of the Z direction
		9	-	X component of the X direction
		10	-	Y component of the X direction

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		11	-	Z component of the X direction
		12	-	Type of angle definition:
		13	-	Angle 1
		14	-	Angle 2
		15	-	Angle 3

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Tool usage and tooling				
	975	1	-	Tool usage test for the current program: Result –2: Test not possible, function disabled in the configuration Result –1: Test not possible, tool usage file missing Result 0: Test OK, all tools available Result 1: Test not OK
		2	Line	Check availability of the tools required in the pallet from line IDX in the current pallet table. –3 = No pallet is defined in row IDX, or function was called outside of pallet editing –2 / –1 / 0 / 1 see NR1
Lift off the tool at NC stop				
	980	3	-	(This function is obsolete—HEIDENHAIN recommends not to use it any longer. ID980 NR3 = 1 is equivalent to ID980 NR1 = –1, ID980 NR3 = 0 has the same effect as ID980 NR1 = 0. Other values are not permissible.) Enable lift-off to the value defined in CfgLiftOff: 0 = Lock lift-off function 1 = Enable lift-off function
Touch probe cycles and coordinate transformations				
	990	1	-	Approach behavior: 0 = Standard behavior 1 = Approach probing position without compensation Effective radius, set-up clearance is zero
		2	16	Automatic / Manual machine operating modes
		4	-	0 = Stylus not deflected 1 = Stylus deflected
		6	-	TT tool touch probe active? 1 = Yes 0 = No
		8	-	Momentary spindle angle in [°]
		10	QS parameter no.	Determine the tool number from the tool name. The return value depends on the rules configured for the search of the replacement tool. If there are multiple tools with the same name, the first tool from the tool table will be selected. If the tool selected by these rules is locked, a replacement tool will be returned. –1: No tool with the specified name found in the tool table or all qualifying tools are locked.

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		16	0	0 = Transfer control over the channel spindle to the PLC, 1 = Assume control over the channel spindle
			1	0 = Pass tool spindle control to the PLC, 1 = Take control of the tool spindle
		19	-	Suppress touch prove movement in cycles: 0 = Movement will be suppressed (CfgMachineSimul/simMode parameter not equal to FullOperation or **Test Run** operating mode is active) 1 = Movement will be performed (CfgMachineSimul/simMode parameter = FullOperation, can be programmed for testing purposes)
Status of execution				
	992	10	-	Block scan active 1 = yes, 0 = no
		11	-	Block scan—information on block scan: 0 = Program started without block scan 1 = Iniprog system cycle is run before block scan 2 = Block scan is running 3 = Functions are being implemented –1 = Iniprog cycle was canceled before block scan –2 = Cancelation during block scan –3 = Cancelation of the block scan after the search phase, before or during the update of functions –99 = Implicit cancelation
		12	-	Type of canceling for interrogation within the OEM_CANCEL macro: 0 = No cancellation 1 = Cancellation due to error or emergency stop 2 = Explicit cancellation with internal stop after stop in the middle of the block 3 = Explicit cancellation with internal stop after stop at the end of a block
		14	-	Number of the last FN14 error
		16	-	Real execution active? 1 = execution, 0 = simulation
		17	-	2-D graphics during programming active? 1 = yes 0 = no
		18	-	Generate graphics during programming (soft key **AUTO DRAW**) active? 1 = yes 0 = no

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
		20	-	Information on combined milling/turning mode of operation: 0 = Milling (after **FUNCTION MODE MILL**) 1 = Turning (after **FUNCTION MODE TURN**) 10 = Execute the operations for the turning-to-milling transition 11 = Execute the operations for the milling-to-turning transition
		30	-	Interpolation of multiple axes permitted? 0 = No (e.g. for straight cut control) 1 = yes
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes
		32	0	Cycle call possible/permitted? 0 = No 1 = Yes
			Cycle number	Single cycle enabled: 0 = No 1 = Yes
		40	-	Copy tables in **Test Run** operating mode? Value 1 will be set when a program is selected and when the **RESET+START** soft key is pressed. The **iniprog.h** system cycle will then copy the tables and reset the system datum. 0 = no 1 = yes
		101	-	M101 active (visible condition)? 0 = no 1 = yes
		136	-	M136 active? 0 = no 1 = yes

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Activate machine parameter subfile				
	1020	13	QS parameter no.	Has a machine parameter subfile with path from QS number (IDX) been loaded? 1 = Yes 0 = No
Configuration settings for cycles				
	1030	1	-	Display **spindle does not rotate** error message? **(CfgGeoCycle/displaySpindleErr)** 0 = no, 1 = yes
			-	Check **the algebraic sign for depth error message!** display? **(CfgGeoCycle/displayDepthErr)** 0 = no, 1 = yes
Write or read PLC data synchronously in real time				
	2000	10	Marker no.	PLC markers General note for NR10 to NR80: The functions are executed synchronously in real time, i.e. the function is not executed until the corresponding point is reached in the program. HEIDENHAIN recommends using the **WRITE TO PLC** or **READ FROM PLC** commands instead of ID2000 and synchronizing the execution in real time by using **FN20: WAIT FOR SYNC**.
		20	Input no.	PLC input
		30	Output no.	PLC output
		40	Counter no.	PLC counter
		50	Timer no.	PLC timer
		60	Byte no.	PLC byte
		70	Word no.	PLC word
		80	Double-word no.	PLC double word

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Do not write or read PLC data synchronously in real time				
	2001	10-80	see ID 2000	Same as ID2000 NR10 to NR80, but not synchronous in real time. Function is executed in the look-ahead calculation. HEIDENHAIN recommends using the **WRITE TO PLC** and **READ FROM PLC** commands instead of ID2001.
Bit test				
	2300	Number	Bit number	This function checks whether a bit has been set in a number. The number to be checked is transferred as NR, the bit to be searched for as IDX, with IDX0 designating the least significant bit. To call this function for great numbers, make sure to transfer NR as a Q parameter. 0 = Bit not set 1 = Bit set
Read program information (system string)				
	10010	1	-	Path of the pallet subprogram, without subprogram calls using **CALL PGM**
		3	-	Path of the cycle selected with **SEL CYCLE** or **CYCLE DEF 12 PGM CALL**, or path of the currently active cycle
		10	-	Path of the NC program selected with **SEL PGM "..."**.
Read channel data (system string)				
	10025	1	-	Name of machining channel (key)
Read data for SQL tables (system string)				
	10040	1	-	Symbolic name of the preset table.
		2	-	Symbolic name of the datum table.
		3	-	Symbolic name of the pallet preset table.
		10	-	Symbolic name of the tool table.
		11	-	Symbolic name of the pocket table.
Read machine kinematics				
	10290	10	-	Symbolic name of the machine kinematics from Channels/ChannelSettings/CfgKinList/kinCompositeModels programmed in **FUNCTION MODE MILL** or **FUNCTION MODE TURN**.
Read data of touch probes (TS, TT) (system string)				
	10350	50	-	TS probe type from TYPE column of the touch probe table (**tchprobe.tp**)
		70	-	Type of TT tool touch probe from CfgTT/type.
		73	-	Key name of the active tool touch probe TT from **CfgProbes/activeTT**.

Group name	Gruppen-nummerID	Systemdaten-nummer	Index	Description
Read and write data of touch probes (TS, TT) (system string)				
	10350	74	-	Serial number of the active tool touch probe TT from **CfgProbes/activeTT**.
Read the data for pallet processing (system string)				
	10510	1	-	Pallet name.
		2	-	Path of the selected pallet table.
Read version ID of the NC software (system string)				
	10630	10	-	This string corresponds to the format of the version ID displayed, i.e. **340590 07** or **817601 04 SP1**.
Read data of the current tool (system string)				
	10950	1	-	Current tool name.

Example: Assign the value of the active scaling factor for the Z axis to Q25.

```
55 FN 18: SYSREAD Q25 = ID210 NR4 IDX3
```

FN 19: PLC – Transfer values to the PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

The **FN 19: PLC** function transfers up to two numerical values or Q parameters to the PLC.

FN 20: WAIT FOR – NC and PLC synchronization

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

With the **FN 20: WAIT FOR** function you can synchronize the NC and PLC during a program run. The NC stops machining until the condition that you have programmed in the **FN 20: WAIT FOR** block is fulfilled.

SYNC is used whenever you read, for example, system data via **FN 18: SYSREAD** that require synchronization with real time. The control stops the look-ahead calculation and executes the following NC block only when the NC program has actually reached that block.

Example: Pause internal look-ahead calculation, read current position in the X axis

```
32 FN 20: WAIT FOR SYNC
33 FN 18: SYSREAD Q1 = ID270 NR1 IDX1
```

FN 29: PLC – Transfer values to the PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

The **FN 29: PLC** function transfers up to eight numerical values or Q parameters to the PLC.

FN 37: EXPORT

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

You need the **FN 37: EXPORT** function if you want to create your own cycles and integrate them in the control.

FN 38: SEND – Send information from NC program

The function **FN 38: SEND** enables you to write texts and Q parameter values to the log from the NC program and send to a DNC application.

Data transmission is through a standard TCP/IP computer network.

For more detailed information, consult the Remo Tools SDK manual.

Example

Document values from Q1 and Q23 in the log.

```
FN 38: SEND /"Q parameter Q1: %f Q23: %f" / +Q1 / +Q23
```

9.9 Accessing tables with SQL commands

Introduction

If you would like to access numerical or alphanumerical content in a table or manipulate the table (e.g., rename columns or rows), then use the SQL commands available to you.

The syntax of the SQL commands available on the control is heavily influenced by the SQL programming language—but does not conform to it completely. In addition, the control does not support the entire scope of the SQL language.

The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when inputting data or reading it out.

The following terms will be used (along with others) in the following:

- "SQL command" refers to the available soft keys
- "SQL instructions" describe miscellaneous functions that are entered manually as part of the syntax
- In the syntax, **HANDLE** stands for a transaction (followed by the identifying parameter)
- The **Result set** contains the query result (referred to in the following as "intermediate memory")

Read and write access to individual numerical values of a table can likewise be carried out using the function **FN 26: TABOPEN**, **FN 27: TABWRITE** and **FN 28: TABREAD**.
Further information: "Freely definable tables", page 351

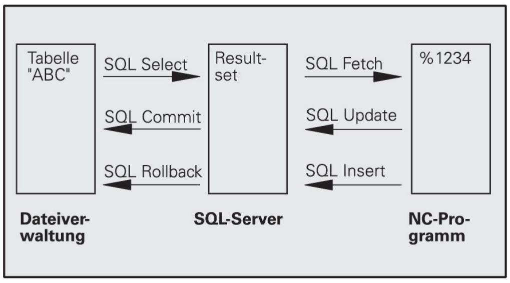
In the NC software, access to tables is gained via an SQL server. This server is controlled with the available SQL commands. The SQL commands can be defined directly in an NC program.

The saver is based on a transaction model. A **transaction** is made up of multiples steps that are executed together, thereby ensuring an orderly and defined processing of the table entries.

Transaction

Example of an SQL transaction:

- Assign Q parameters to table columns for read or write access using **SQL BIND**
- Select data using **SQL SELECT** or **SQL EXECUTE** with the **SELECT** instruction
- Read, change, or add data using **SQL FETCH**, **SQL UPDATE**, and **SQL INSERT**
- Confirm or discard interaction using **SQL COMMIT** and **SQL ROLLBACK**
- Approve bindings between table columns and Q parameters using **SQL BIND**



> **i** You must conclude all transactions that have been started—even exclusively read accesses. Concluding the transaction is the only way to ensure that changes and additions are transferred, that locks are removed, and that used resources are released.

Overview of functions

Overview of soft keys

Soft key	Command	Page
SQL BIND	**SQL BIND** establishes or removes connections between table columns and Q or QS parameters	304
SQL EXECUTE	**SQL EXECUTE** opens a transaction for selected table columns and table rows or enables the use of other SQL instructions (miscellaneous functions). **Further information:** "Overview of instructions", page 302	305
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	308
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	312
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	311
SQL UPDATE	**SQL UPDATE** transfers all values from the bound Q parameters to the table	309
SQL INSERT	**SQL INSERT** creates a new table row	310
SQL SELECT	**SQL SELECT** reads out a single values from a table and does not open any transaction	313

Overview of instructions

The following so-called SQL instructions are used in the SQL command **SQL EXECUTE**.

Further information: "SQL EXECUTE", page 305

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate a table
COPY TABLE	Copying a table
RENAME TABLE	Rename table
DROP TABLE	Delete the table
INSERT	Inserting table rows
DELETE	Delete table rows
ALTER TABLE	■ Add table columns using **ADD** ■ Delete table columns using **DROP**
RENAME COLUMN	Rename table columns

Programming SQL commands

> This function is not enabled until the code number **555343** is entered.

You can program SQL commands in the **Programming** operating mode or in **Positioning with mdi**:

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

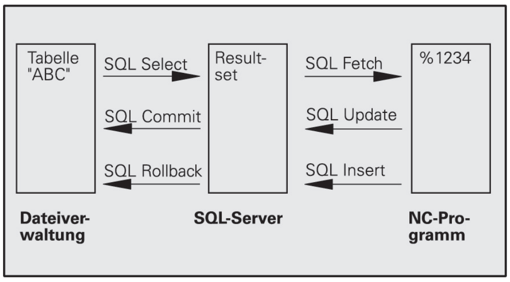
▷
- Shift the soft-key row

SQL
- Press the **SQL** soft key
- Select the SQL command via soft key

> Read and write accesses performed with the help of SQL commands always occur in metric units, regardless of the unit of measure selected for the table or the NC program.
>
> If, for example, a length is saved from one table to a Q parameter, then the value is thereafter always in metric units. If this value is then use in an inch program for the purpose of positioning (**L X+Q1800**), then an incorrect position will be the result.

Application example

In the following example, the defined material will be read out from the table (**MILL.TAB**) and saved as text in a QS parameter. The following example shows a possible application and the necessary program steps.



> You can continue to use texts from QS parameters in separate log files, for example, by using the function **FN16**.
>
> **Further information:** "FN16: F-PRINT – Formatted output of texts and Q parameter values", page 261

Example

0 BEGIN PGM SQL MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\MILL.TAB'"	Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NO==3"	Define search
4 SQL FETCH Q1900 HANDLE QL1	Execute search
5 SQL ROLLBACK Q1900 HANDLE QL1	Complete transaction
6 SQL BIND QS1800	Remove parameter binding
7 SQL Q1 "DROP SYNONYM my_table"	Delete synonym
8 END PGM SQL MM	

Step	Explanation
1 Create synonym	A synonym is assigned to a path (long path names are replaced by short names) ■ The path **TNC:\table\MILL.TAB** must contained in single quotation marks for this. ■ The selected synonym is **my_table**
2 Bind QS parameters	A QS parameter is bound to a table column ■ **QS1800** is freely available in user programs ■ The synonym replaces the entry of the complete path ■ The defined column from the table is called **WMAT**
3 Define search	A search definition contains the entry of the transfer value ■ The **QL1** local parameter (freely selectable) serves to identify the transaction (multiple transactions are possible simultaneously) ■ The synonym defines the table ■ The **WMAT** entry defines the table column of the read operation ■ The entries **NO** and **=3** define the table rows of the read operation ■ Selected table columns and rows define the cells of the read operation

4 Execute search	The read operation is executed
	■ The **Q1900** parameter is only important for the transaction (return value if needed for checking)
	  ■ **0** successful read operation
	  ■ **1** faulty read operation
	■ The **HANDLE QL1** syntax is the transaction designated by the **QL1** parameter
	■ The value is copied from the so-called **result set** (intermediate memory) to the bound parameter

Step	Explanation
5 Complete transaction	The transaction is concluded and the used resources are released
6 Remove binding	The binding between table columns and QS parameters is removed (release of necessary resources)
7 Delete synonym	The synonym is deleted again (release of necessary resources)

SQL BIND

Example: binding Q parameters to table columns

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
```

Example: remove binding

```
91 SQL BIND Q881
92 SQL BIND Q882
93 SQL BIND Q883
94 SQL BIND Q884
```

SQL BIND links a Q parameter to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) for the data transfer between the **result set** (intermediate memory) and the NC program.

An **SQL BIND** command without a table or column name cancels the link. The link is terminated at the end of the NC program or subprogram, if not before.

> Programming notes:
> - You can program any number of bindings. During read and write operations, the only columns taken into consideration are those that are specified using the **SELECT** command. If you specify columns without binding in the **SELECT** command, then the control will interrupt the read or write operation with an error message.
> - **SQL BIND...** must be programmed **before** the **FETCH**, **UPDATE**, and **INSERT** commands.

SQL BIND

- **Parameter no. for result**: define Q parameter for binding to the table column
- **Database: column name**: define table name and table column (separate with **.**)
  - **Table name**: synonym or path with filename of the table
  - **Column name**: name displayed in the table editor

SQL EXECUTE

SQL EXECUTE is used in connection with various SQL instructions.**Further information:** "Overview of instructions", page 302

SQL EXECUTE with the SQL instruction SELECT

The SQL server places the data in rows in the **result set** (intermediate memory). The rows are numbered in ascending order, starting from 0. This row number (the **INDEX**) is used for the SQL commands **FETCH** and **UPDATE**.

SQL EXECUTE, in combination with the SQL instruction **SELECT**, selects table values and transfers them to the **result set**. In contrast to the SQL command **SQL SELECT**, the combination of **SQL EXECUTE** and the instruction **SELECT** selects multiple columns and rows simultaneously and always opens a transaction.

In the function **SQL ... “SELECT...WHERE..."**, you can enter the search criteria. This lets you restrict the number of rows to be transferred. If you do not use this option, then all of the rows in the table are loaded.

In the function **SQL ... “SELECT...ORDER BY..."**, you can enter the ordering criterion. This entry consists of the column designation and the keyword (**ASC**) for ascending or (**DESC**) for descending order. If you do not use this option, then rows will be stored in a random order.

With the function **SQL ... "SELECT...FOR UPDATE"**, you can lock the selected rows for other applications. Other applications can continue to read these rows but are unable to change them. If you make changes to the table entries, then it is absolutely necessary to use this option.

Empty result set: If none of the rows correspond to the search criteria, then the SQL software returns a valid **HANDLE** (transaction) but not any table entries.

Example: selection of table rows

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
    Tab_Example"
```

Example: selection of table rows with the WHERE function

```
. . .
20 SQL Q5 "SELECT Meas_No,Meas_X,Meas_Y, Meas_Z FROM
    Tab_Example WHERE Meas_No<20"
```

Example: selection of table rows with the WHERE function and Q parameters

```
. . .
20 SQL Q5 "SELECT Meas_No,Meas_X,Meas_Y, Meas_Z FROM
    Tab_Example WHERE Meas_No==:’Q11’"
```

Example: table name defined with path and file name

```
. . .
20 SQL Q5 "SELECT Meas_No,Meas_X,Meas_Y, Meas_Z FROM 'V:
   \table\Tab_Example' WHERE Meas_No<20"
```

SQL EXECUTE

- **Parameter No. for result** (return value for the control):
  - **0** successful read operation
  - **1** faulty read operation
- **Database: SQL command text**: programming SQL instruction
  - **SELECT** with the table column(s) to be transferred (separate multiple columns with **,**)
  - **FROM** with a table's synonym or path (place the path in single quotation marks)
  - **WHERE** (optional) with column names, condition, and comparison value (Q parameters after **:** in single quotation marks)
  - **ORDER BY** (optional) with column names and type of ordering (**ASC** for ascending, **DESC** for descending order)
  - **FOR UPDATE** (optional) to lock write access to the selected row for other processes

Conditions for WHERE entries

Condition	Programming
Equals	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
empty	IS NULL
Not empty	IS NOT NULL
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

Syntax examples

The following examples are listed without context. The NC blocks are limited exclusively to the possibilities of the SQL command **SQL EXECUTE**.

Example

9 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\MILL.TAB'"	Create synonym
9 SQL Q1800 "DROP SYNONYM my_table"	Delete synonym
9 SQL Q1800 "CREATE TABLE my_table (NO,WMAT)"	Create table with the rows NO and WMAT.
9 SQL Q1800 "COPY TABLE my_table TO 'TNC:\table\MILL2.TAB'"	Copy table
9 SQL Q1800 "RENAME TABLE my_table TO 'TNC:\table\MILL3.TAB'"	Rename table
9 SQL Q1800 "DROP TABLE my_table"	Delete the table
9 SQL Q1800 "INSERT INTO my_table VALUES (1,'ENAW',240)"	Insert table row
9 SQL Q1800 "DELETE FROM my_table WHERE NO==3"	Delete table row
9 SQL Q1800 "ALTER TABLE my_table ADD (WMAT2)"	Insert table rows
9 SQL Q1800 "ALTER TABLE my_table DROP (WMAT2)"	Delete table rows
9 SQL Q1800 "RENAME COLUMN my_table (WMAT2) TO (WMAT3)"	Rename table column

SQL FETCH

Example: transferring row number in the Q parameter

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: programming the row number directly

```
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX5
```

SQL FETCH reads a row out of the **result-set** (intermediate memory). The values of the individual cells are stored in the bound Q parameters. The transaction is defined via the **HANDLE** to be specified; the row is defined via the **INDEX**.

SQL FETCH takes all columns into consideration that were specified with the **SELECT** instruction (SQL command **SQL EXECUTE**).

SQL FETCH

- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: define Q parameters for the **HANDLE** (for identifying the transaction)
- **Database: index to SQL result**: row number within the **result set**
  - Program the row number directly
  - Program the Q parameter containing the index
  - The row (n=0) is read if nothing is specified

The optional syntax elements **IGNORE UNBOUND** and **UNDEFINE MISSING** are intended for the machine tool builder.

SQL UPDATE

Example: transferring row number in the Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MESS_NR"
12 SQL BIND Q882 "TAB_EXAMPLE.MESS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MESS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MESS_Z"
. . .
20 SQL Q5 "SELECT MESS_NR,MESS_X,MESS_Y,MESS_Z FROM
    TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: programming the row number directly

```
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

SQL UPDATE changes a row in the **result set** (intermediate memory). The new values of the individual cells are copied from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified; the row is defined via the **INDEX**. The existing row in the **result set** is completely overwritten.

SQL UPDATE takes all columns into consideration that were specified with the **SELECT** instruction (SQL command **SQL EXECUTE**).

- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)
- **Database: Index for SQL result**: Row number within the **result set**
  - Program the row number directly
  - Program the Q parameter containing the index
  - The row (n=0) is assigned a value if none is specified

SQL INSERT

Example: Transferring row number in the Q parameter

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
    Tab_Example"
. . .
40 SQL INSERT Q1 HANDLE Q5
```

SQL UPDATE creates a new row in the **result set** (intermediate memory). The values of the individual cells are copied from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified.

SQL INSERT takes all columns into consideration that were specified with the **SELECT** instruction (SQL command **SQL EXECUTE**). Table columns without corresponding **SELECT** instruction (not contained in the query result) are assigned defaults values.

SQL INSERT

- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)

SQL COMMIT

Example

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL COMMIT Q1 HANDLE Q5
```

SQL COMMIT simultaneously transfers all of the rows that have been changed and added in a transaction back into the table. The transaction is defined via the **HANDLE** to be specified. A lock that was set with **SELECT...FOR UPDATE** is canceled.

The **HANDLE** (process) assigned with the instruction **SQL SELECT** becomes invalid.

SQL COMMIT

- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)

SQL ROLLBACK

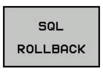
Example

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL ROLLBACK Q1 HANDLE Q5
```

SQL ROLLBACK discards all of the changes and additions of a transaction. The transaction is defined via the **HANDLE** to be specified.

The function of the SQL command **SQL ROLLBACK** depends on the **INDEX**:

- Without **INDEX**:
  - All changes and additions to the transaction are discarded
  - A lock that was set with **SELECT...FOR UPDATE** is canceled.
  - The transaction is concluded (the **HANDLE** loses its validity)
- With **INDEX**:
  - Only the indexed row remains in the **result set** (all other rows are removed)
  - Any changes and additions made in the rows that are not specified are discarded
  - A lock that has been set with **SELECT...FOR UPDATE** remains only for indexed row (all other locks are canceled)
  - The specified (indexed) row becomes the new row 0 of the **result-set**
  - The transaction is **not** concluded (the **HANDLE** keeps its validity)
  - It is necessary to later concluded the transaction using **SQL ROLLBACK** or **SQL COMMIT**



- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)
- **Database: Index to SQL result**: Row that remains in the **result set**
  - Program the row number directly
  - Program the Q parameter containing the index

SQL SELECT

SQL SELECT reads a single value from a table and saves the result in the defined Q parameter.

> You can select multiple values or columns using the SQL command **SQL EXECUTE** and the **SELECT** instruction.
> **Further information:** "SQL EXECUTE", page 305

With **SQL SELECT**, there is neither a transaction nor binding between the table columns and Q parameter. Any existing bindings to the specified columns are not taken into consideration; only the read-out value is copied into the parameter specified for the result.

Example: Reading and saving a value

```
20 SQL SELECT Q5 "SELECT Meas_X FROM Tab_Example WHERE
   MEAS_NO==3"
```

SQL SELECT

- **Parameter No. for result**: Q parameter for saving the value
- **Database: SQL command text**: Programming SQL instruction
  - **SELECT** with the table column of the value to be transferred
  - **FROM** with a table's synonym or path (place the path in single quotation marks)
  - **WHERE** with column designation, condition and comparison value (Q parameter after **:** in single quotation marks)

The result of the subsequent NC program is identical to the application example shown previously.
Further information: "Application example", page 303

Example

0 BEGIN PGM SQL MM	
1 SQL SELECT QS1800 "SELECT WMAT FROM my_table WHERE NO==3"	Read and save a value
2 END PGM SQL MM	

9.10 Entering formulas directly

Entering formulas

Using soft keys, you can enter mathematical formulas containing multiple calculation operations directly into the NC program.

Q
- Select Q-parameter functions

FORMULA
- Press the **FORMULA** soft key
- Select **Q**, **QL**, or **QR**

The control displays the following soft keys in several soft-key rows:

Soft key	Linking function
+	**Addition** e.g., **Q10 = Q1 + Q5**
-	**Subtraction** e.g., **Q25 = Q7 - Q108**
*	**Multiplication** e.g., **Q12 = 5 * Q5**
/	**Division** e.g., **Q25 = Q1 / Q2**
(	**Opening parenthesis** e.g., **Q12 = Q1 * (Q2 + Q3)**
)	**Closing parenthesis** e.g., **Q12 = Q1 * (Q2 + Q3)**
SQ	**Square the value** e.g., **Q15 = SQ 5**
SQRT	**Calculate square root** e.g., **Q22 = SQRT 25**
SIN	**Sine of an angle** e.g., **Q44 = SIN 45**
COS	**Cosine of an angle** e.g., **Q45 = COS 45**
TAN	**Tangent of an angle** e.g., **Q46 = TAN 45**
ASIN	**Arc sine** Inverse function of the sine; determine the angle from the ratio of the opposite side to the hypotenuse e.g., **Q10 = ASIN 0.75**
ACOS	**Arc cosine** Inverse function of the cosine; determine the angle from the ratio of the adjacent side to the hypotenuse e.g., **Q11 = ACOS Q40**

Soft key	Linking function
ATAN	**Arc tangent** Inverse function of the tangent; determine the angle from the ratio of the opposite side to the adjacent side e.g., **Q12 = ATAN Q50**
^	**Powers of values** e.g., **Q15 = 3^3**
PI	**Constant PI (3,14159)** e.g., **Q15 = PI**
LN	**Calculate the natural logarithm of a number** Base 2,7183 e.g., **Q15 = LN Q11**
LOG	**Logarithm of a number, Base 10** e.g., **Q33 = LOG Q22**
EXP	**Exponential function, 2.7183 to the power of n** e.g., **Q1 = EXP Q12**
NEG	**Negate values (multiply by -1)** e.g., **Q2 = NEG Q1**
INT	**Remove digits after the decimal point** Calculate an integer e.g., **Q3 = INT Q42**
ABS	**Absolute value of a number** e.g., **Q4 = ABS Q22**
FRAC	**Remove digits before the decimal point** Calculate a fraction e.g., **Q5 = FRAC Q23**
SGN	**Check algebraic sign of a number** e.g., **Q12 = SGN Q50** When return value Q12 = 0, then Q50 = 0 When return value Q12 = 1, then Q50 > 0 When return value Q12 = -1, then Q50 < 0
%	**Calculate modulo value (division remainder)** e.g., **Q12 = 400 % 360** result: Q12 = 40

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first

Example

```
12  Q1 = 5 * 3 + 2 * 10 = 35
```

1 Calculation 5 * 3 = 15
2 Calculation 2 * 10 = 20
3 Calculation 15 + 20 = 35

or

Example

```
13 Q2 = SQ 10 - 3^3 = 73
```

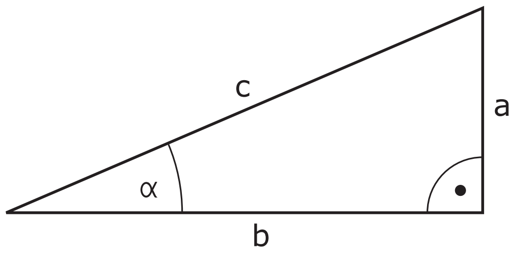
1 Calculation step 10 squared = 100
2 Calculation step 3 to the third power = 27
3 Calculation 100 – 27 = 73

Distributive law

Law of distribution with parentheses calculation
a * (b + c) = a * b + a * c

Example of entry

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.



Q

FORMULA

- Select the formula entry function: Press the **Q** key and the **FORMULA** soft key

Q

- Press the **Q** key on the external ASCII keyboard

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **25** (parameter number) and press the **ENT** key

▷

ATAN

- Shift the soft-key row and select the arc tangentfunction

◁

(

- Advance through the soft key menu and press the **OPENING PARENTHESIS** soft key

Q

- Enter **12** (Q parameter number)

/

- Select division

Q

- Enter **13** (Q parameter number)

)

END

- Close parentheses and conclude formula entry

Example

```
37 Q25 = ATAN (Q12/Q13)
```

9.11 String parameters

String processing functions

You can use the **QS** parameters to create variable character strings. You can output such character strings for example through the **FN 16:F-PRINT** function to create variable logs.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) up to a length of 255 characters to a string parameter. You can also check and process the assigned or imported values using the functions described below. As in Q parameter programming, you can use a total of 2000 QS parameters.

Further information: "Principle and overview of functions", page 242

The **STRING FORMULA** and **FORMULA** Q parameter functions contain various functions for processing the string parameters.

Soft key	Functions of the STRING FORMULA	Page
STRING	Assigning string parameters	319
CFGREAD	Read out machine parameter	328
	Chain-linking string parameters	319
TOCHAR	Converting a numerical value to a string parameter	321
SUBSTR	Copy a substring from a string parameter	322
SYSSTR	Read system data	323

Soft key	Formula string functions	Page
TONUMB	Converting a string parameter to a numerical value	324
INSTR	Checking a string parameter	325
STRLEN	Finding the length of a string parameter	326
STRCOMP	Compare alphabetic priority	327

When you use the **STRING FORMULA** function, 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.

Assign string parameters

Before using string variables, you must first assign the variables. Use the **DECLARE STRING** command to do so.

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

STRING FUNCTIONS
- Press the **STRING FUNCTIONS** soft key

DECLARE STRING
- Press the **DECLARE STRING** soft key

Example

```
37 DECLARE STRING QS10 = "Workpiece"
```

Chain-linking string parameters

With the concatenation operator (string parameter **||** string parameter) you can make a chain of two or more string parameters.

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

STRING FUNCTIONS
- Press the **STRING FUNCTIONS** soft key

STRING FORMULA
- Press the **STRING FORMULA** soft key

ENT
- Enter the number of the string parameter in which the control 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 control shows the concatenation symbol **||** an.
- Press 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

```
37 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 control converts a numerical value into a string parameter. This enables you to chain numerical values with string variables.

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Open the function menu

STRING FUNCTIONS
- Press the String functions soft key

STRING FORMULA
- Press the **STRING FORMULA** soft key

TOCHAR
- Select the function for converting a numerical value to a string parameter
- Enter the number or the desired Q parameter to be converted by the control, and confirm with the **ENT** key
- If desired, enter the number of digits after the decimal point that the control 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

```
37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```

Copying a substring from a string parameter

The **SUBSTR** function copies a definable range from a string parameter.

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Open the function menu

STRING FUNCTIONS
- Press the String functions soft key

STRING FORMULA
- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the character 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

> The first character of a text string starts internally at the 0-position

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2)

```
37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```

Reading system data

With the function **SYSSTR** you can read system data and store them in string parameters. You select the system data through a group number (ID) and a number.

Entering IDX and DAT is not required.

Group name, ID no.	Number	Meaning
Program information, 10010	1	Path of the current main program or pallet program
	3	Path of the cycle selected with **CYCL DEF 12 PGM CALL**
	10	Path of the program selected with **SEL PGM**
Channel data, 10025	1	Channel name
Values programmed in the tool call, 10060	1	Tool name
Current system time, 10321	1 - 16	■ 1: DD.MM.YYYY hh:mm:ss ■ 2 and 16: DD.MM.YYYY hh:mm ■ 3: DD.MM.YY hh:mm ■ 4: YYYY-MM-DD hh:mm:ss ■ 5 and 6: YYYY-MM-DD hh:mm ■ 7: YY-MM-DD hh:mm ■ 8 and 9: DD.MM.YYYY ■ 10: DD.MM.YY ■ 11: YYYY-MM-DD ■ 12: YY-MM-DD ■ 13 and 14: hh:mm:ss ■ 15: hh:mm
Touch-probe data, 10350	50	Probe type of the active touch probe TS
	70	Probe type of the active touch probe TT
	73	Key name of the active touch probe TT from MP **activeTT**
	2	Path of the selected pallet table
NC software version, 10630	10	Version identifier of the NC software version
Tool data, 10950	1	Tool name
	2	DOC entry of the tool
	4	Tool-carrier kinematics

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 to be converted must contain only one numerical value. Otherwise, the Control will output an error message..

Q

- Select Q-parameter functions

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the string parameter in which the control 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 QS parameter to be converted by the control, 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

```
37 Q82 = TONUMB ( SRC_QS11 )
```

Testing a string parameter

The **INSTR** function checks whether a string parameter is contained in another string parameter.

Q

- Select Q-parameter functions

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter for the result and confirm with the **ENT** key
- > The control saves the place at which the text to be searched for begins. It is saved in the parameter.

◁

- Shift the soft-key row

INSTR

- Select the function for checking a string parameter
- Enter the number of the QS parameter in which the text to be searched for is saved. Confirm with the **ENT** key
- Enter the number of the QS parameter to be searched for by the control, and confirm with the **ENT** key
- Enter the number of the place at which the control is to start 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

> The first character of a text string starts internally at the 0-position
>
> If the control cannot find the required substring, it will save the total length of the string to be searched (counting starts at 1) in the result parameter.
>
> If the substring to be searched for appears multiple times, then the control 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.

```
37 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 function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control 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 from which the control is to ascertain the length, 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

```
37 Q52 = STRLEN ( SRC_QS15 )
```

> If the selected string parameter is not defined the control returns the result **-1**.

Comparing alphabetic priority

The **STRCOMP** function compares string parameters for alphabetic priority.

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the result of comparison, and 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 that the control is to compare, and confirm with the **ENT** key
- Enter the number of the second QS parameter that the control is to compare, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

The control 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

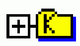
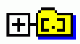
Example: QS12 and QS14 are compared for alphabetic priority

```
37 Q52 = STRCOMP  ( SRC_QS12 SEA_QS14 )
```

Reading out machine parameters

With the **CFGREAD** function, you can read out machine parameters of the control as numerical values or as strings. The read-out values are always output in metric units of measure.

In order to read out a machine parameter, you must use the control's configuration editor to determine the parameter name, parameter object, and, if they have been assigned, the group name and index:

Icon	Type	Meaning	Example
	**Key**	Group name of the machine parameter (if available)	CH_NC
	**Entity**	Parameter object (name begins with **Cfg**...)	**CfgGeoCycle**
	Attribute	Name of the machine parameter	**displaySpindleErr**
	**Index**	List index of a machine parameter (if available)	[0]

> If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts.
>
> **Further information:** "Changing the display of the parameters", page 640

Each time you want to interrogate a machine parameter with the **CFGREAD** function, you must first define a QS parameter with attribute, entity and key.

The following parameters are read in the CFGREAD function’s dialog:

- **KEY_QS**: Group name (key) of the machine parameter
- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

Reading a string of a machine parameter

In order to store the content of a machine parameter as a string in a QS parameter:

Q

- Press the **Q** key.

STRING FORMULA

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the machine parameter
- Press the **ENT** key
- Select the **CFGREAD** function
- Enter the numbers of the string parameters for key, entity, and attribute
- Press the **ENT** key
- Enter the number for the index, or skip the dialog with N**NO ENT**, whichever applies
- Close the parenthesized expression with the **ENT** key
- Press the **END** key to conclude entry

Example: Read as a string the axis designation of the fourth axis

Parameter settings in the configuration editor

DisplaySettings

CfgDisplayData

axisDisplayOrder

[0] to [3]

Example

14 QS11 = ""	Assign string parameter for key
15 QS12 = "CfgDisplaydata"	Assign string parameter for entity
16 QS13 = "axisDisplay"	Assign string parameter for parameter name
17 QS1 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13 IDX3)	Read out machine parameter

Reading a numerical value of a machine parameter

Store the value of a machine parameter as a numerical value in a Q parameter:

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the machine parameter
- Press the **ENT** key
- Select the **CFGREAD** function
- Enter the numbers of the string parameters for key, entity, and attribute
- Press the **ENT** key
- Enter the number for the index, or skip the dialog with N**NO ENT**, whichever applies
- Close the parenthesized expression with the **ENT** key
- Press the **END** key to conclude entry

Example: Read overlap factor as Q parameter

Parameter settings in the configuration editor

ChannelSettings

CH_NC

CfgGeoCycle

pocketOverlap

Example

14 QS11 = "CH_NC"	Assign string parameter for key
15 QS12 = "CfgGeoCycle"	Assign string parameter for entity
16 QS13 = "pocketOverlap"	Assign string parameter for parameter name
17 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	Read out machine parameter

9.12 Preassigned Q parameters

The Q parameters Q100 to Q199 are assigned values by the control. The following types of information are assigned to the Q parameters:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Results of measurements from touch probe cycles etc.

The control saves the preassigned Q parameters Q108, Q114 and Q115 - Q117 in the unit of measure used by the active program.

NOTICE

Danger of collision!

Q parameters are used in the HEIDENHAIN cycles, in machine tool builder cycles, and in supplier functions. You can also program Q parameters within the NC program. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.
- Check the machining sequence using a graphic simulation

You must not use preassigned Q parameters (QS parameters) between **Q100** and **Q199** (**QS100** and **QS199**) as calculation parameters in the NC programs.

Values from the PLC: Q100 to Q107

The control assigns values from the PLC to parameters Q100 to Q107 in an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or **TOOL DEF** block)
- Delta value DR from the tool table
- Delta value DR from the **TOOL CALL** block

The control remembers the current tool radius even if the power is interrupted.

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 the parameter Q110 depends on the M function last programmed for the spindle.

M function	Parameter value
No spindle status defined	Q110 = –1
M3: Spindle ON, clockwise	Q110 = 0
M4: Spindle ON, counterclockwise	Q110 = 1
M5 after M3	Q110 = 2
M5 after M4	Q110 = 3

Coolant on/off: Q111

M function	Parameter value
M8: Coolant ON	Q111 = 1
M9: Coolant OFF	Q111 = 0

Overlap factor: Q112

The control assigns Q112 to the overlap factor for pocket milling.

Unit of measurement for dimensions in the program: Q113

During nesting the **PGM CALL**, the value of the parameter Q113 depends on the dimensional data of the program from which the other programs are called.

Dimensional data of the main program	Parameter value
Metric system (mm)	Q113 = 0
Imperial system (inch)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

> The Control remembers the current tool length even if the power is interrupted.

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 refer to the preset that is active in the **Manual operation** mode.

The length of the stylus and the radius of the ball tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117
4th axis Machine-dependent	Q118
5th axis Machine-dependent	Q119

Deviation between actual value and nominal value during automatic tool measurement with, for example, the TT 160

Deviation of actual from nominal value	Parameter value
Tool length	Q115
Tool radius	Q116

10

Miscellaneous Functions

10.1 Enter miscellaneous functions M

Fundamentals

With the control’s miscellaneous functions—also called M functions—you can affect:

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

You can enter up to four M (miscellaneous) functions at the end of a positioning block or in a separate block. The control displays the following dialog question: **Miscellaneous function M ?**

You usually enter only the number of the miscellaneous function in the programming dialog. Some miscellaneous 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** operating modes, the M functions are entered with the **M** soft key.

Effectiveness of miscellaneous functions

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

> If multiple functions were programmed in a single NC block, the execution sequence is as follows:
>
> - M functions taking effect at the start of the block are executed before those taking effect at the end of the block
> - If all M functions are effective at the start or end of the block, execution takes place in the sequence as programmed

Entering a miscellaneous function in a STOP block

If you program a **STOP** block, the program run or test run is interrupted at the block, e.g. for a tool inspection. You can also enter an M (miscellaneous) function in a **STOP** block:

STOP

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

Example

```
87 STOP M6
```

10.2 Miscellaneous functions for program run inspection, spindle and coolant

Overview

> Refer to your machine manual.
> The machine manufacturer can influence the behavior of the miscellaneous functions described below.

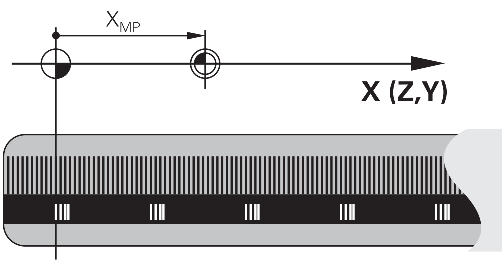
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope depends on machine parameter **resetAt** (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M6	Tool change Spindle STOP Program STOP			■
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

10.3 Miscellaneous functions for coordinate entries

Programming machine-referenced coordinates: M91/ M92

Scale datum

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



Machine datum

The machine datum is required for the following tasks:

- Define the axis traverse limits (software limit switches)
- Approach machine-referenced positions (e.g. tool change positions)
- Set a workpiece preset

The distance in each axis from the scale datum to the machine datum is defined by the machine manufacturer in a machine parameter.

Standard behavior

The control references the coordinates to the workpiece datum.

Further information: "Presetting without a 3-D touch probe", page 403

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 control screen reference the machine datum. Switch the display of coordinates in the status display to REF.

Further information: "Status displays", page 85

Behavior with M92 – Additional machine reference point

Refer to your machine manual.

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

For each axis, the machine tool builder defines the distance between the machine reference point and the machine datum.

If you want the coordinates in positioning blocks to be based on the additional machine reference point, end these block with M92.

Radius compensation remains the same in blocks that are programmed with **M91** or **M92**. The tool length will **not** be taken into account.

Effect

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

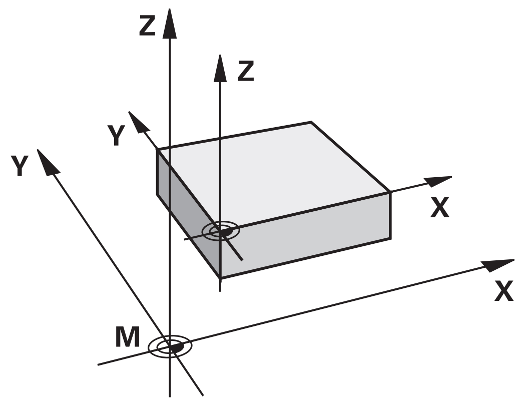
M91 and M92 take effect at the start of block.

Workpiece preset

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

If presetting is inhibited for all axes, the control no longer displays the **SET PRESET** soft key in the **Manual operation** mode.

The figure shows coordinate systems with the machine 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 defined preset.

Further information: "Showing the workpiece blank in the working space", page 442

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

Standard behavior

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

If you entered a traverse limit or a software limit switch is active, **M94** is ineffective for the corresponding axis.

Example: Reduce the display of all active rotary axes

```
M94
```

Example: Reduce the display of the C axis

```
M94 C
```

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

```
C+180  FMAX M94
```

Effect

M94 is effective only in the NC block where it is programmed.

M94 becomes effective at the start of the block.

10.4 Miscellaneous functions for path behavior

Feed rate factor for plunging movements: M103

Standard behavior

The control moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103

The control 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 program **M103** in a positioning block, the control continues the dialog by prompting you for the F factor.

Effect

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

Feed rate in millimeters per spindle revolution: M136

Standard behavior

The control moves the tool at the programmed F feed rate in mm/min

Behavior with M136

> In NC programs based on inch units, **M136** is not allowed in combination with the alternative **FU** feed rate.
>
> The spindle is not permitted to be controlled when M136 is active.

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

Effect

M136 becomes effective at the start of the block.

You can cancel **M136** by programming **M137**.

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

Standard behavior

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control moves the tool as defined in the machining program.

Behavior with M140

With **M140 MB** (move back), you can retract the tool from the contour by a programmable distance in the direction of the tool axis.

Input

If you enter **M140** in a positioning block, the control continues the dialog and prompts you for the path the tool should use for retracting from the contour. Enter the desired path that the tool should follow when retracting from the contour, or press the **MB 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 control moves the tool along the entered path at rapid traverse.

Effect

M140 is effective only in the NC block in which itis programmed.

M140 becomes effective at the start of the block.

Example

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

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

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

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

Always define a tool call with tool axis before **M140**, otherwise the traverse direction is not defined.

11

Special Functions

11.1 Overview of special functions

The control provides the following powerful special functions for a large number of applications:

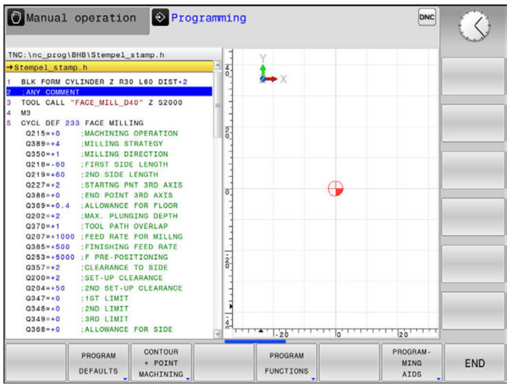
Function	Description
Working with text files	page 364
Working with freely definable tables	page 351

Press the **SPEC FCT** key and the corresponding soft keys to access further special functions of the control. The following tables give you an overview of which functions are available.

Main menu for SPEC FCT special functions

SPEC FCT

- Press the **SPEC FCT** key to select the special functions



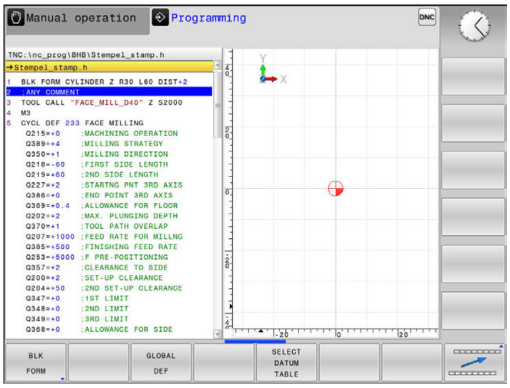
Soft key	Function	Description
PROGRAM DEFAULTS	Define program defaults	page 347
CONTOUR + POINT MACHINING	Functions for contour and point machining	page 347
PROGRAM FUNCTIONS	Define different conversational functions	page 348
PROGRAM-MING AIDS	Programming aids	page 153

After pressing the **SPEC FCT** key, you can open the **smartSelect** selection window with the **GOTO** key. The control displays a structure overview with all available functions. You can rapidly navigate with the cursor or mouse and select functions in the tree diagram. The control displays online help for the selected function in the window on the right.

Program defaults menu

PROGRAM DEFAULTS

- Press the Program Defaults soft key

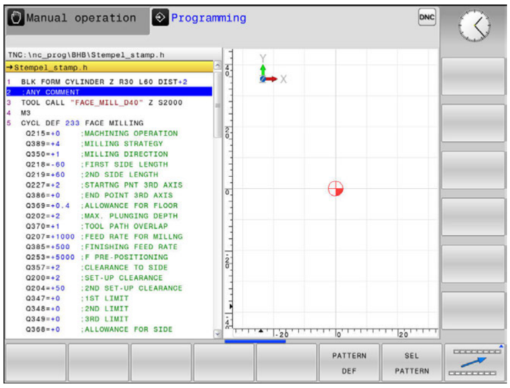


Soft key	Function	Description
BLK FORM	Define workpiece blank	page 113
DATUM TABLE	Select datum table	page 596
GLOBAL DEF	Define global cycle parameters	page 506

Functions for contour and point machining menu

CONTOUR + POINT MACHINING

- Press the soft key for functions for contour and point machining

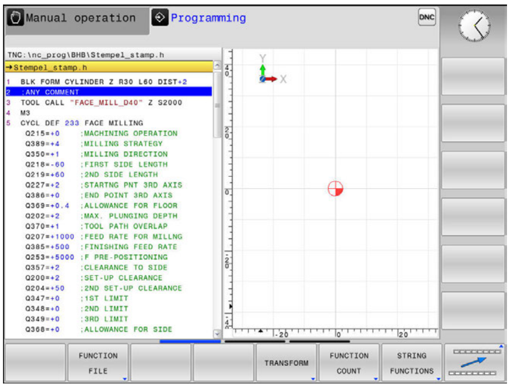


Soft key	Function	Description
PATTERN DEF	Define regular machining pattern	page 509
SEL PATTERN	Select the point file with machining positions	page 520

Menu for defining different conversional functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key



Soft key	Function	Description
FUNCTION FILE	Define file functions	page 360
TRANSFORM	Define coordinate transformations	page 361
FUNCTION COUNT	Define the counter	page 349
STRING FUNCTIONS	Define string functions	page 318
FUNCTION SPINDLE	Define pulsing spindle speed	page 356
FUNCTION FEED	Define recurring dwell time	page 358
FUNCTION DWELL	Define dwell time in seconds or revolutions	page 373
INSERT COMMENT	Add comments	page 155

11.2 Defining a counter

Application

> Refer to your machine manual.
> Your machine manufacturer enables this function.

The FUNCTION COUNT function allows you to control a simple counter from within the NC program. For example, this function allows you to count the number of manufactured workpieces. The counter is only effective in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.

The counter values are retained even after a restart of the control.

You can use Cycle 225 to engrave the current counter value into the workpiece.

Proceed as follows for the definition:

SPEC FCT
- Show the soft key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION COUNT
- Press the **FUNCTION COUNT** soft key

NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.

- Please check prior to machining whether a counter is active.
- If necessary, note down the counter value and enter it again via the MOD menu after execution.

Define FUNCTION COUNT

The **FUNCTION COUNT** function provides the following possibilities:

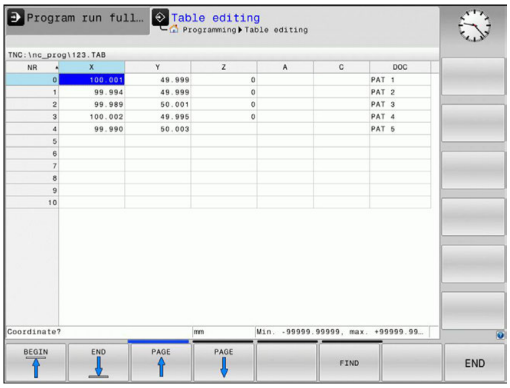
Soft key	Meaning
FUNCTION COUNT INC	Increase count by 1
FUNCTION COUNT RESET	Reset counter
FUNCTION COUNT TARGET	Set the nominal count (target value) to the desired value Input value: 0–9999
FUNCTION COUNT SET	Set the counter to the desired value Input value: 0–9999
FUNCTION COUNT ADD	Increment the counter by the desired value Input value: 0–9999
FUNCTION COUNT REPEAT	Repeat the NC program starting from this label if more parts are to be machined.

Example

5 FUNCTION COUNT RESET	Reset the counter value
6 FUNCTION COUNT TARGET10	Enter the target number of parts to be machined
7 LBL 11	Enter the jump label
8 ...	Machining
51 FUNCTION COUNT INC	Increment the counter value
52 FUNCTION COUNT REPEAT LBL 11	Repeat the machining operations if more parts are to be machined.
53 M30	
54 END PGM	

11.3 Freely definable tables

Fundamentals



In freely definable tables you can save and read any information from the NC program. The Q parameter functions **FN 26** to **FN 28** are provided for this purpose.

You can change the format of freely definable tables, i.e. the columns and their properties, by using the structure editor. They enable you to make tables that are exactly tailored to your application.

You can also toggle between a table view (standard setting) and form view.

> The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when inputting data or reading it out.

Creating a freely definable table

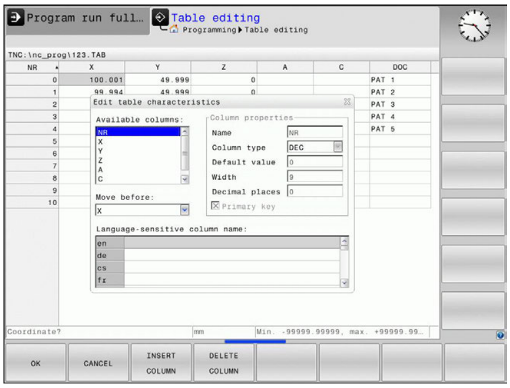
- To call the file manager, press the **PGM MGT** key
- Enter any desired file name with the .TAB extension and confirm it with the **ENT** key
- > The control displays a pop-up window with permanently stored table formats
- Use the arrow key to select the desired table template, e.g. **example.tab** and confirm it with the **ENT** key
- > The control opens a new table in the predefined format
- To adapt the table to your requirements you have to edit the table format
  Further information: "Editing the table format", page 352

> Refer to your machine manual.
> Machine tool builders may define their own table templates and save them in the control. When you create a new table, the control opens a pop-up window listing all available table templates.

> You can also save your own table templates in the control. To do so, create a new table, change the table format and save the table in the **TNC:\system\proto** directory. Then your template will also be available in the list box for table templates when you create a new table.

Editing the table format

- Press the **EDIT FORMAT** soft key (toggle the soft key row)
- > The control opens the editor form displaying the table structure. The meanings of the structure commands (header entries) are shown in the following table.



Structure command	Meaning
Available columns:	List of all columns contained in the table
Move before:	The entry highlighted in **Available columns** is moved in front of this column
Name	Column name: Is displayed in the header
Column type	**TEXT**: Text entry **SIGN**: + or - sign **BIN**: Binary number **DEC**: Decimal, positive, whole number (cardinal number) **HEX**: Hexadecimal number **INT**: Whole number **LENGTH**: Length (is converted in inch programs) **FEED**: Feed rate (mm/min or 0.1 inch/min) **IFEED**: Feed rate (mm/min or inch/min) **FLOAT**: Floating-point number **BOOL**: Logical value **INDEX**: Index **TSTAMP**: Fixed format for date and time **UPTEXT**: Text entry in upper case **PATHNAME**: Path name
Default value	Default value for the fields in this column
Width	Width of the column (number of characters)
Primary key	First table column
Language-sensitive column name	Language-sensitive dialogs

Use a connected mouse or the control's keyboard to navigate in the form. Navigation using the control's keyboard:

- Press the navigation keys to go to the entry fields. Use the arrow keys to navigate within an entry field. To open pop-down menus, press the **GOTO** key.

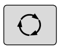
> In a table that already contains lines you can not change the table properties **Name** and **Column type**. Once you have deleted all lines, you can change these properties. If required, create a backup copy of the table beforehand.
>
> With the **CE** and **ENT** key combination, you can reset invalid values in fields with the **TSTAMP** column type.

Exiting the structure editor

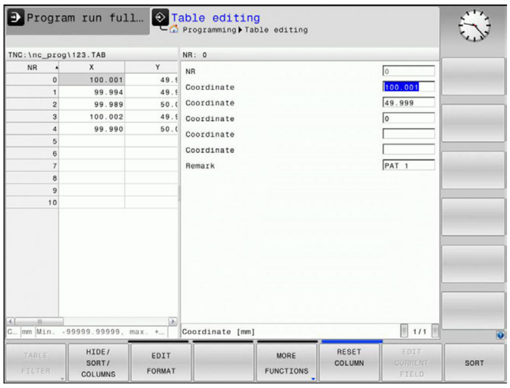
- Press the **OK** soft key
- > The control closes the editor form and applies the changes. All changes are discarded by pressing the **CANCEL** soft key.

Switching between table and form view

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



- Press the key for setting the screen layout. Select the respective soft key for list view or form view (form view: with or without dialog texts)



In the left half of the form view, the control lists the line numbers with the contents of the first column.

In the right half you can change the data.

- Press the **ENT** key or the arrow key to move to the next entry field
- To select another line press the navigation key (folder symbol). This moves the cursor to the left window, and you can select the desired line with the arrow keys. Press the green navigation key to switch back to the input window.

FN 26: TABOPEN – Open a freely definable table

With the function **FN 26: TABOPEN** you open a freely definable table to be written to with **FN 27** or to be read from with **FN 28**.

> Only one table can be opened in an NC program at any one time. A new block with **FN 26: TABOPEN** automatically closes the last opened table.
>
> The table to be opened must have the extension **.TAB**.

Example: Open the table TAB1.TAB, which is saved in the directory TNC:\DIR1.

```
56 FN 26: TABOPEN TNC:\DIR1\TAB1.TAB
```

FN 27: TABWRITE – Write to a freely definable table

With the **FN 27: TABWRITE** function you write to the table that you previously opened with **FN 26: TABOPEN**.

You can define multiple column names in a **TABWRITE** block. The column names must be written between quotation marks and separated by a comma. You define in Q parameters the value that the control is to write to the respective column.

> The **FN 27: TABWRITE** function by default writes values to the currently open table, even in the **Test Run** operating mode. The **FN 18 ID992 NR16** function allows you to retrieve the operating mode in which the program is running. If the **FN27** function is to be run only in the **Program run, single block** and **Program run, full sequence** operating modes, you can skip the respective program section by using a jump statement.
>
> **Further information:** "If-then decisions with Q parameters", page 252
>
> You can write only to numerical table fields.
>
> If you wish to write to more than one column in a block, you must save the values under successive Q parameter numbers.

Example

You wish to write to the columns “Radius,” “Depth” and “D” in line 5 of the presently opened table. The value to be written in the table must be saved in the Q parameters Q5, Q6 and Q7.

```
53 Q5 = 3.75
54 Q6 = -5
55 Q7 = 7.5
56 FN 27:  TABWRITE 5/"RADIUS,DEPTH,D" = Q5
```

FN 28: TABREAD – Read from a freely definable table

With the **FN 28: TABREAD** function you read from the table previously opened with **FN 26: TABOPEN**.

You can define, i.e. read, multiple column names in a **TABREAD** block. The column names must be written between quotation marks and separated by a comma. In the **FN 28** block you can define the Q parameter number in which the control is to write the value that is first read.

> You can read only numerical table fields.
>
> If you wish to read from more than one column in a block, the control will save the values under successive Q parameter numbers.

Example

You wish to read the values of the columns "Radius," "Depth" and "D" from line 6 of the presently opened table. Save the first value in Q parameter Q10 (second value in Q11, third value in Q12).

```
56 FN 28:  TABREAD Q10 = 6/"RADIUS,DEPTH,D"
```

Customizing the table format

NOTICE

Caution: Data may be lost!

The **ADAPT NC PGM / TABLE** function changes the format of all tables permanently. Existing data is not automatically backed up by the control before running the format change process, i.e. the files are changed permanently and might no longer be usable.

- Only use the function in consultation with the machine tool builder.

Soft key	Function
ADAPT NC PGM / TABLE	Adapt format of tables present after changing the control software version

> The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when inputting data or reading it out.

11.4 Pulsing spindle speed FUNCTION S-PULSE

Programming a pulsing spindle speed

Application

> Refer to your machine manual.
> Read and note the functional description of the machine tool builder.
> Follow the safety precautions.

Using the **S-PULSE FUNCTION** you can program a pulsing spindle speed, when operating at a constant spindle speed.

You can define the duration of a vibration (period length) using the P-TIME input value or a speed change in percent using the SCALE input value. The spindle speed changes in a sinusoidal form around the target value.

Procedure

Example

```
13 FUNCTION S-PULSE P-TIME10 SCALE5
```

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION SPINDLE
- Press the **FUNCTION SPINDLE** soft key

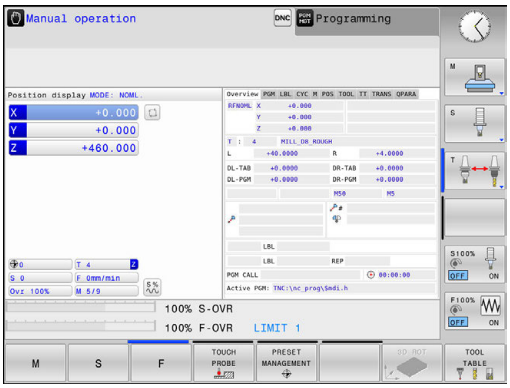
SPINDLE-PULSE
- Press the **SPINDLE-PULSE** soft key
- Define period length P-TIME
- Define speed change SCALE

> The control never exceeds a programmed speed limit. The spindle speed is maintained until the sinusoidal curve of the **S-PULSE FUNCTION** falls below the maximum speed once more.

Symbols

In the status bar the symbol indicates the condition of the pulsing shaft speed:

Icon	Function
S %	Pulsing spindle speed active



Resetting the pulsing spindle speed

Example

```
18 FUNCTION S-PULSE RESET
```

Use the **FUNCTION S-PULSE RESET** to reset the pulsing spindle speed.

Proceed as follows for the definition:

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION SPINDLE

- Press the **FUNCTION SPINDLE** soft key

RESET SPINDLE-PULSE

- Press the **RESET SPINDLE-PULSE** soft key.

11.5 Dwell time FUNCTION FEED

Programming dwell time

Application

Refer to your machine manual.

Read and note the functional description of the machine tool builder.

Follow the safety precautions.

The **FUNCTION FEED DWELL** function can be used to program a recurring dwell time in seconds, e.g. to force chip breaking . Program **FUNCTION FEED DWELL** immediately prior to the machining you wish to run with chip breaking.

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motion.

NOTICE

Caution: Danger to the tool and workpiece!

When the **FUNCTION FEED DWELL** function is active, the control will repeatedly interrupt the feed movement. While the feed movement is interrupted, the tool remains at its current position while the spindle continues to turn. Due to this behavior, workpieces need to be scrapped if threads are cut. In addition, there is a danger of tool breakage during execution!

- Deactivate the **FUNCTION FEED DWELL** function before cutting threads

Procedure

Example

```
13 FUNCTION FEED DWELL D-TIME0.5 F-TIME5
```

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION FEED
- Press the **FUNCTION FEED** soft key

FEED DWELL
- Press the **FEED DWELL** soft key
- Define the interval duration for dwelling D-TIME
- Define the interval duration for cutting F-TIME

Resetting dwell time

> Reset to the dwell time immediately following the machining with chip breaking.

Example

```
18 FUNCTION FEED DWELL RESET
```

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

Proceed as follows for the definition:

SPEC FCT	▶ Show the soft-key row with special functions
PROGRAM FUNCTIONS	▶ Press the **PROGRAM FUNCTIONS** soft key
FUNCTION FEED	▶ Press the **FUNCTION FEED** soft key
RESET FEED DWELL	▶ Press the **RESET FEED DWELL** soft key

> You can also reset the dwell time by entering D-TIME 0.
> The control automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

11.6 File functions

Application

The **FILE FUNCTION** functions are used to perform file operations such as copying, moving, and deleting files from within the NC program.

> You must not use **FILE** functions on programs or files, to which you have previously made reference with functions such as **CALL PGM** or **CYCL DEF 12 PGM CALL**.

Defining file functions

SPEC FCT
- Press the special functions key

PROGRAM FUNCTIONS
- Select the program functions

FUNCTION FILE
- Select file operations
- > The control displays the available functions.

Soft key	Function	Meaning
FILE COPY	**FILE COPY**	Copy file: Enter the name and path of the file to be copied, as well as the target path
FILE MOVE	**FILE MOVE**	Move file: Enter the name and path of the file to be moved, as well as the target path
FILE DELETE	**FILE DELETE**	Delete file: Enter the path and name of the file to be deleted

If you try to copy a file that does not exist, the control generates an error message.

FILE DELETE does not generate an error message if you try to delete a non-existing file.

11.7 Defining coordinate transformations

Overview

As an alternative to the coordinate transformation Cycle 7, **DATUM SHIFT**, you can also use the **TRANS DATUM** conversational function. Just as in Cycle 7, you can use **TRANS DATUM** to directly program shift values or activate a line from a selectable datum table. In addition, there is also the **TRANS DATUM RESET** function that can be used to easily reset a datum shift.

TRANS DATUM AXIS

Example

```
13 TRANS DATUM AXIS X+10 Y+25 Z+42
```

You can define a datum shift by entering values in the respective axis with the **TRANS DATUM AXIS** function. You can define up to nine coordinates in one block, and incremental entries are possible. Proceed as follows for the definition:

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

TRANSFORM

- Select transformations

TRANS DATUM

- Select the **TRANS DATUM** datum shift

VALUES X Y Z

- Select the value input soft key
- Enter the datum shift in the affected axes, confirming with the **ENT** key each time

> Values entered as absolute numbers refer to the workpiece preset, which is specified either by presetting or by selecting a preset from the preset table.
>
> Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted).

TRANS DATUM TABLE

Example

13 TRANS DATUM TABLE TABLINE25

You can define a datum shift by selecting a datum number from a datum table with the **TRANS DATUM TABLE** function. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

TRANSFORM
- Select transformations

TRANS DATUM
- Select the **TRANS DATUM** datum shift

TABLE .D
- Select the **TRANS DATUM TABLE** datum shift
- Enter the line number to be activated by the control, confirm with the **ENT** key
- If desired, enter the name of the datum table from which you want to activate the datum number, and confirm with the **ENT** key. If you do not want to define a datum table, confirm with the **NO ENT** key

> If you have not defined a datum table in the **TRANS DATUM TABLE** block, then the control uses the datum table previously selected with **SEL TABLE** or the datum table activated in the **Program run, single block** or **Program run, full sequence** operating mode (status **M**).

TRANS DATUM RESET

Example

```
13 TRANS DATUM  RESET
```

Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

TRANSFORM
- Select transformations

TRANS DATUM
- Select the **TRANS DATUM** datum shift

RESET DATUM SHIFT
- Press the **RESET DATUM SHIFT** soft key

11.8 Creating text files

Application

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

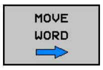
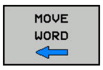
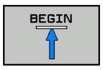
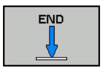
- Recording test results
- Documenting working procedures
- Creating formula collections

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

Opening and exiting a text file

- Operating mode: Press the **Programming** key
- To call the file manager, press the **PGM MGT** key.
- Display type .A files: Press the **SELECT TYPE** soft key and **SHOW ALL** soft key one after the other
- 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.

Soft key	Cursor movements
	Move cursor one word to the right
	Move cursor one word to the left
	Cursor at beginning of file
	Cursor at end of file

Editing texts

Above the first line of the text editor, there is an information field showing the file name, location and line information:

File:	Name of the text file
Line:	Line in which the cursor is presently located
Column:	Column in which the cursor is presently located

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.

You can insert a line break with the **RETURN** or **ENT** key.

Deleting and re-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 deleted and stored temporarily.
- Move the cursor to the location where you wish insert the text, and press the **INSERT LINE / WORD** soft key.

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

Editing text blocks

You can copy and erase text blocks of any size, and insert them at other locations. Before any of these actions, 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:

Soft key	Function
CUT OUT BLOCK	Delete the selected block and store temporarily
COPY BLOCK	Store the selected block temporarily without erasing (copy)

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—the text block is inserted.

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

Transferring the selected block to a different file

- Select the text block as described previously

APPEND TO FILE

- Press the **APPEND TO FILE** soft key.
- > The control displays the **file name** dialog message.
- Enter the path and the name of the destination file.
- > The control appends the selected text block to the specified file.

Inserting 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 control displays the **File name =** dialog message.
- 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. The control provides the following two options.

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 find a word: press the **FIND** soft key.
- Exit the search function: Press the **END** soft key

Finding any text

- To select the search function, press the **FIND** soft key. The control shows the **Find text :** dialog prompt
- Enter the text that you wish to find
- To find text: press the **FIND** soft key.
- Exit the search function: Press the **END** soft key

11.9 Tool carrier management

Fundamentals

You can create and manage tool carriers using the tool carrier management. The control factors the tool carriers into the calculations.

On machines with 3 axes, tool carriers for right-angled angled heads help processing on tool axes **X** and **Y**, as the control takes the dimensions of the angle heads into consideration.

You must carry out the following steps so that the control can factors the tool carriers into the calculations:

- Save tool carrier templates
- Assign input parameters to tool carriers
- Allocate parameterized tool carriers

Save tool carrier templates

Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. So that you don't have to design all your tool carriers yourself, HEIDENHAIN supplies a range of ready-made tool carrier templates. Tool carrier templates are 3-D models with fixed geometries but changeable dimensions.

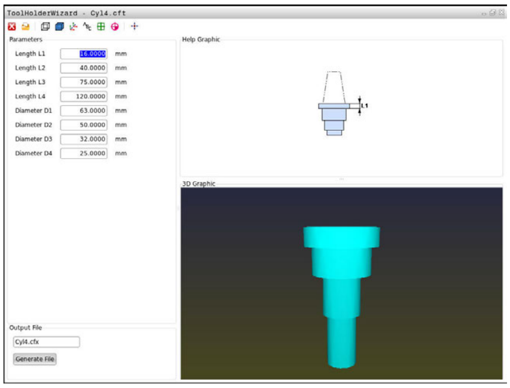
The tool carrier templates must be saved in **TNC:\system \Toolkinematics** and have the extension **.cft**.

> If the tool carrier templates are not available in your control, please download the data you require from:
> **http://www.klartext-portal.com/nc-solutions/en**

> If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

> The tool carrier templates may consist of several sub-files. If the sub-files are incomplete, the control will display an error message.
> **Do not use incomplete tool carrier templates!**

Assigning input parameters to tool carriers

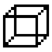
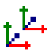
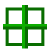
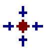


Before the control can factor the tool carrier into the calculations, you must give the tool carrier template the actual dimensions. These parameters are entered in the additional **ToolHolderWizard** tool.

Save the parameterized tool carriers with the extension **.cfx** under **TNC:\system\Toolkinematics**.

The additional **ToolHolderWizard** tool is mainly operated with a mouse. Using the mouse, you can also set the desired screen layout by drawing a line between the areas **Parameter**, **Help graphics** and **3-D graphics** by holding down the left mouse button.

The following icons are available in the additional **ToolHolderWizard** tool:

Icon	Function
	Close tool
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Display or hide transformation vectors
	Show or hide names of collision objects
	Display or hide test points
	Show or hide measurement points
	Return to starting view of the 3-D model

> **i** If the tool carrier template does not contain any transformation vectors, names, test points and measurement points, the additional **ToolHolderWizard** tool does not execute any function when the corresponding icons are activated.

Parameterizing the tool carrier template in the Manual operation operating mode

Proceed as follows to parameterize tool carrier templates and save these parameters:

- Press the **Manual operation** key

- Press the **TOOL TABLE** soft key

- Press the **EDIT** soft key

- Move the cursor to the **KINEMATIC** column

- Press the **SELECT** soft key

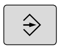
- Press the **TOOL HOLDER WIZARD** soft key
- > The control opens the additional **ToolHolderWizard** tool in a pop-up window.

- Press the **OPEN FILE** icon
- > The control opens a pop-up window.
- Select the desired tool carrier template using the preview screen
- Press the **OK** button
- > The control opens the selected tool carrier template.
- > The cursor goes to the first parameterizable value.
- Adjust values
- Enter the name for the parameterized tool holder in the **Output file** area
- Press the **GENERATE FILE** button
- If required, reply to the message on the control

- Press the **CLOSE** icon
- > The control closes the additional tool

Parameterizing the tool carrier template in the Programming operating mode

Proceed as follows to parameterize tool carrier templates and save these parameters:



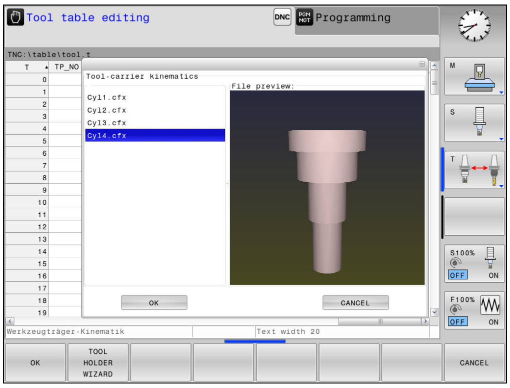
- Press the **Programming** key

- Press the **PGM MGT** key
- Select the path **TNC:\system\Toolkinematics**
- Select the tool carrier template
- > The control opens the additional **ToolHolderWizard** tool with the selected tool carrier template.
- > The cursor goes to the first parameterizable value.
- Adjust values
- Enter the name for the parameterized tool holder in the **Output file** area
- Press the **GENERATE FILE** button
- If required, reply to the message on the control

- Press the **CLOSE** icon
- > The control closes the additional tool

Allocating parameterized tool carriers

To allow the control to factor a parameterized tool carrier into calculations, you must allocate the tool carrier to a tool and **call the tool again**.



> **i** Parameterized tool carriers can consist of several sub-files. If the sub-files are incomplete, the control will display an error message.
>
> **Only use fully parameterized tool carriers!**

Proceed as follows to allocate a parameterized tool carrier to a tool:

- ▶ Operating mode: Press the **Manual operation** key

- ▶ Press the **TOOL TABLE** soft key

- ▶ Press the **EDIT** soft key

- ▶ Move the cursor to the **KINEMATIC** column of the required tool

- ▶ Press the **SELECT** soft key
- > The control opens a pop-up window with parameterized tool carriers
- ▶ Select the desired tool carrier using the preview screen
- ▶ Press the **OK** soft key
- > The control copies the name of the selected tool carrier to the **KINEMATIC** column

- ▶ Exit the tool table

11.10 Dwell time FUNCTION DWELL

Programming dwell time

Application

The **FUNCTION DWELL** function enables you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

Procedure

Example

```
13 FUNCTION DWELL TIME10
```

Example

```
23 FUNCTION DWELL REV5.8
```

Proceed as follows for the definition:

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION DWELL

- **FUNCTION DWELL** soft key

DWELL TIME

- Press the **DWELL TIME** soft key
- Define the duration in seconds

DWELL REVOLUTIONS

- Alternatively, press the **DWELL REVOLUTIONS** soft key
- Define the number of spindle revolutions

12

Manual Operation and Setup

12.1 Switch-on, switch-off

Switch-on

DANGER

Caution: Danger for the operator!

Machines and machine components always present mechanical hazards. Electric, magnetic or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- Read and follow the machine manual
- Read and follow the safety precautions and safety symbols
- Use the safety devices

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

Switch the machine and the control on as follows:

- Switch on the power supply for the control and the machine
- > The control displays the switch-on status in the subsequent dialogs.
- > If booting was successful, the control displays the **Power interrupted** dialog

CE

- Press the **CE** key to clear the message
- > The control displays the **Compiling PLC program** dialog; the PLC program is compiled automatically
- > The control displays the **Switch on external dc voltage** dialog

I

- Switch on the machine control voltage
- > The control carries out a self-test.

If the control does not register an error, it displays the **Traverse reference points** dialog.

If the control registers an error, it issues an error message.

Check the axis positions

> This section applies only to machine axes with EnDat encoders.

If the actual axis position after the machine is switched on does not match the position at switch-off, the control displays a pop-up window.

- Check the axis position of the affected axis
- If the current axis position matches that proposed in the display, confirm with **YES**

NOTICE

Danger of collision!

If they are not paid attention to, deviations between the actual axis positions and those expected by the control (saved at the time of switch-off) can lead to undesirable and unforeseeable movements of the axes. There is risk of collision during referencing and all subsequent movements.

- Check the axis positions
- Only confirm the pop-up window with **YES** if the axis positions match
- Despite confirmation, at first only move the axis carefully
- If there are discrepancies or you have any doubts, contact your machine tool builder

Traverse reference points

If the control performs the self-test successfully, it then displays the **Traverse reference points** dialog.

> Refer to your machine manual.
>
> Switching on the machine and traversing the reference points can vary depending on the machine tool.
>
> If your machine is equipped with absolute encoders, you can leave out crossing the reference points.

> If you intend only to edit or graphically simulate NC programs, you can select the **Programming** or **Test Run** mode of operation immediately after switching on the control voltage, without needing to reference the axes.
>
> You can neither set a preset nor modify a preset via the preset table without having referenced the axes. The control issues the **Traverse reference points** hint.
>
> You can cross the reference points later. For this purpose, in **Manual operation** mode press the **PASS OVER REFERENCE** soft key.

Cross the reference points manually in the displayed sequence:

- For each axis press the **NC START** button, or
- > The control is now ready for operation in the **Manual operation** mode.

As an alternative you can cross the reference points in any sequence:

X+

Y+

- Press and hold the axis direction button for each axis until the reference point has been traversed
- > The control is now ready for operation in the **Manual operation** mode.

Switch-off

	Refer to your machine manual. Deactivation is a machine-dependent function.

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

- Operating mode: Press the **Manual operation** key

OFF

- Press the **OFF** soft key

SHUT DOWN

- Confirm with the **SHUT DOWN** soft key
- When the control displays the message **Now you can switch off** in a pop-up window, you may switch off the power supply to the control

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss not matter what state the control was in.

- Always shut down the control
- Only turn off the main switch after being prompted on the screen

12.2 Moving the machine axes

Note

	Refer to your machine manual. Movement of the axes via the axis direction keys can vary depending on the machine.

Moving the axis with the axis direction keys

- Operating mode: Press the **Manual operation** key

- Press the axis direction key and hold it down as long as you wish the axis to move; or

- To move the axis continuously: Press and hold the axis direction button and press the **NC Start** key

- To stop: Press the **NC Stop** key

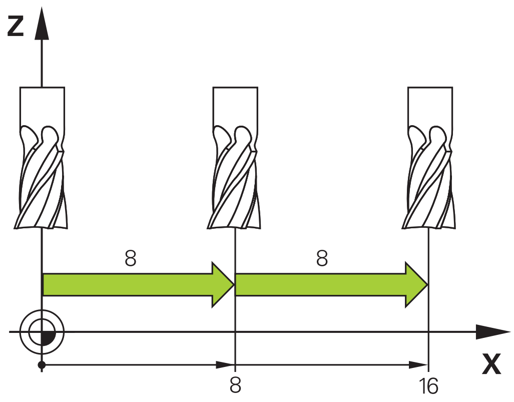
You can change the feed rate at which the axes are moved with the **F** soft key.

Further information: "Spindle speed S, feed rate F and miscellaneous function M", page 394

If a moving task is active on the machine, the control displays the **control in operation** symbol.

Incremental jog positioning

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



- Operating mode: Press the **Manual operation** or **Electronic handwheel** key
- Shift the soft-key row
- Select incremental jog positioning: Switch the **INCREMENT** soft key to **ON**
- Enter the infeed of the **linear axes** and confirm with the **CONFIRM VALUE** soft key
- Alternatively, confirm with the **ENT** key
- Use the arrow keys to position the cursor on the **rotary axis**
- Enter the infeed of the **rotary axes** and confirm with the **CONFIRM VALUE** soft key
- Alternatively, confirm with the **ENT** key
- Confirm with the **OK** soft key
- > The increment is active.
- Deactivate incremental jog positioning: Switch the **INCREMENT** soft key to **OFF**

> If you are in the **Jog increment** menu, you can switch off incremental jog positioning with the **SWITCH OFF** soft key.
>
> The input range for the infeed is from 0.001 mm to 10 mm.

Traverse with the HR 510 electronic handwheel

The portable HR 510 handwheel is equipped with two permissive buttons. The permissive buttons are located above the star grip.

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

The HR 510 handwheel features the following operating elements:

1 EMERGENCY STOP button
2 Handwheel
3 Permissive buttons
4 Axis address keys
5 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)
6 Direction in which the control moves the selected axis
7 Machine functions (set by the machine manufacturer)

Traversing axes

Red LEDs show the active functions, such as the selected axis

- Select the **Electronic handwheel** operating mode
- Press and hold a permissive button
- X Select the axis
- Select the feed rate
- + Move the active axis in the positive direction
- – Move the active axis in the negative direction

Moving with the electronic display handwheels

⚠ DANGER

Caution: Danger for the operator!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- Devices should be connected or removed only by authorized service technicians
- Only switch on the machine via a connected handwheel or a secured connection

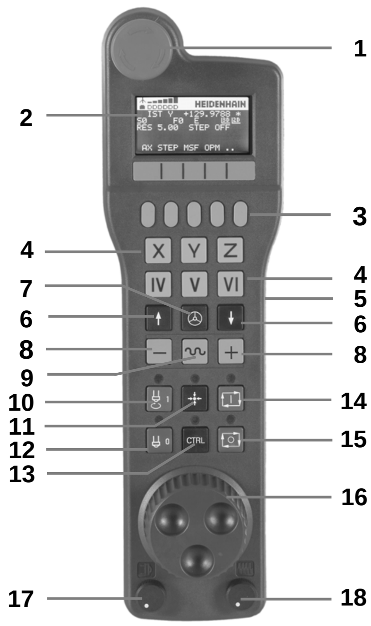
The control supports traversing with the following new electronic handwheels:

- HR 520: Handwheel with display, data transfer via cable
- HR 550FS: Handwheel with display, data transfer via radio

Your machine tool builder can make additional functions of the HR 5xx handwheels available.

The portable HR 520 and HR 550FS handwheels feature a display on which the control shows information. In addition, you can use the handwheel soft keys for important setup functions, e.g. presetting or entering and running M functions.

As soon as you have activated the handwheel with the handwheel activation key, the operating panel is locked. The control shows this status in a pop-up window on the screen.



1 **EMERGENCY STOP** key
2 Handwheel display for status and for selecting functions
3 Soft Keys
4 Axis keys; can be exchanged by the machine manufacturer depending on the axis configuration
5 Permissive button
6 Arrow keys for defining handwheel sensitivity
7 Handwheel activation key
8 Key for traverse direction of the selected axis
9 Rapid traverse superimposing for the axis direction key
10 Spindle switch-on (machine-dependent function, key can be exchanged by the machine manufacturer)
11 **Generate NC block** key (machine-dependent function, key can be exchanged by the machine manufacturer)
12 Spindle switch-off (machine-dependent function, key can be exchanged by the machine manufacturer)
13 **CTRL** key for special functions (machine-dependent function, key can be exchanged by the machine manufacturer)
14 **NC START** key (machine-dependent function, key can be exchanged by the machine manufacturer)
15 **NC STOP** key (machine-dependent function, key can be exchanged by the machine manufacturer)
16 Handwheel
17 Spindle speed potentiometer
18 Feed rate potentiometer
19 Cable connection, not available with the HR 550FS wireless handwheel

Handwheel display

1 **Only with wireless handwheel HR 550FS**: Shows whether the handwheel is in the docking station or whether wireless operation is active

2 **Only with wireless handwheel HR 550FS**: Shows the signal strength, 6 bars = maximum signal strength

3 **Only with wireless handwheel HR 550FS**: Shows the charge status of the rechargeable battery, 6 bars = fully charged A bar moves from the left to the right during recharging

4 **ACTL**: Type of position display

5 **Y+129.9788**: Position of the selected axis

6 *****: STIB (control in operation); program run has been started or axis is in motion

7 **S0:**: Current spindle speed

8 **F0:** Feed rate at which the selected axis is moving

9 **E**: Error message

If an error message appears on the control, the handwheel display shows the message **ERROR** for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.

10 **RES 5.0**: Active handwheel resolution. Path traversed by the selected axis with a handwheel revolution

11 **STEP ON** or **OFF**: Incremental jog active or inactive. If the function is active, the control additionally displays the current traversing step

12 Soft-key row: Selection of various functions, described in the following sections

Special features of the wireless handwheel HR 550FS

⚠ DANGER

Caution: Danger for the operator!

Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.

- Check the radio connection of the handwheel for possible overlapping with other wireless devices
- Switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test when it is restarted
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

Due to various potential sources of interference, a wireless connection is not as reliable as a cable connection. Before a wireless handwheel can be used, it must be checked whether there is an overlapping with other wireless devices. If this is the case, then such overlapping must be eliminated. This inspection for the presence of radio frequencies or channels is obligatory for all industrial radio systems.

If the HR 550 is not needed, always put it in the handwheel holder. This way you can ensure that the handwheel batteries are always ready for use thanks to the contact strip on the rear side of the wireless handwheel and the recharge control, and that there is a direct contact connection for the emergency stop circuit.

If an error (interruption of the radio connection, poor reception quality, defective handwheel component) occurs, the handwheel always reacts with an emergency stop.

The HR 550FS wireless handwheel features a rechargeable battery. The battery starts charging when you put the handwheel in the holder.

You can operate the HR 550FS with the battery for up to 8 hours before it must be recharged again. If not in use, it is recommended to put the handwheel in the handwheel holder.

As soon as the handwheel is in its holder, it switches internally to cable operation. This means you can still use it even if the handwheel is fully discharged. The functions are the same as with wireless operation.

> When the handwheel is completely discharged, it takes about 3 hours until it is fully recharged in the handwheel holder.
>
> Clean the contacts **1** in the handwheel holder and of the handwheel regularly to ensure their proper functioning.

The transmission range is amply dimensioned. If you should nevertheless happen to come near the edge of the transmission area, which is possible with very large machines, the HR 550FS warns you in time with a plainly noticeable vibration alarm. If this happens you must reduce the distance to the handwheel holder in which the radio receiver is integrated.

NOTICE

Caution: Danger to the tool and workpiece!

The wireless handwheel triggers an emergency stop reaction if the radio transmission is interrupted, the battery is fully empty, or if there is a defect. Emergency stop reactions during machining can cause damage to the tool or workpiece.

- Place the handwheel in the handwheel holder when it is not in use
- Keep the distance between the handwheel and the handwheel holder small (pay attention to the vibration alarm)
- Test the handwheel before machining

If the control has triggered an emergency stop you must reactivate the handwheel. Proceed as follows:

- Press the **MOD** key to select the MOD function
- Select **Machine settings**

- Select the configuration menu for the wireless handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- Click the **Start handwheel** button to reactivate the wireless handwheel
- To save the configuration and exit the configuration menu, press **END**

The **MOD** operating mode includes a function for commissioning and configuring the handwheel.

Further information: "Configuring the HR 550FS wireless handwheel", page 494

Selecting the axis to be moved

You can activate the principal axes X, Y, Z and three other axes defined by the machine manufacturer directly through the axis keys. Your machine tool builder can also place the virtual axis VT directly on one of the free axis keys. If the virtual axis VT is not on one of the axis keys, proceed as follows:

- Press the handwheel soft key **F1** (**AX**)
- > The control shows all active axes on the handwheel display. The currently active axis flashes.
- Select the desired axis with the **F1** (**->**) or **F2** soft keys (**<-**) and confirm with the **F3** (**OK**) handwheel soft key

Setting the handwheel sensitivity

The handwheel sensitivity determines which path an axis takes per revolution of the handwheel. The sensitivity levels are pre-defined and are selectable with the handwheel arrow keys (only when incremental jog is not active).

Selectable sensitivity levels:
0.001/0.002/0.005/0.01/0.02/0.05/0.1/0.2/0.5/1 [mm/revolution or degrees/revolution]

Selectable sensitivity levels:
0.00005/0.001/0.002/0.004/0.01/0.02/0.03 [in mm/revolution or degrees/revolution]

Moving the axes

- To activate the handwheel, press the handwheel button on the HR 5xx
- > Now you can operate the control only via the HR 5xx. The control displays a pop-up window with this information on the screen
- Select the desired operating mode with the **OPM** soft key if necessary
- If required, press and hold the permissive button
- Use the handwheel to select the axis to be moved. Select the additional axes with the soft keys as required
- Move the active axis in the positive direction; or
- Move the active axis in the negative direction
- To deactivate the handwheel, press the handwheel key on the HR 5xx
- > Now you can operate the control via the operating panel again.

Potentiometer settings

DANGER

Caution: Danger for the operator!

Activating the handwheel does not automatically activate the potentiometers of the handwheel; rather the potentiometers on the operating panel of the control remain active. After an NC start on the handwheel, the control immediately begins with machining or with axis positioning, even though the potentiometers on the handwheel are set to 0 %. There is a risk of death to anybody inside the working space!

- Before using the handwheel, set the potentiometers of the operating panel to 0 %
- When using the handwheel, always also activate the potentiometers of the handwheel

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 on the HR 5xx at the same time
- > The control shows the soft-key menu for selecting the potentiometers in the handwheel's 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 on the HR 5xx at the same time
- > The control shows the soft-key menu for selecting the potentiometers in the handwheel's display.
- Press the **KBD** soft key to activate the potentiometers of the machine operating panel

The control issues a warning if the handwheel potentiometers are still active after the handwheel has been deactivated.

Incremental jog positioning

With incremental jog positioning the control moves the currently active handwheel axis by a preset increment defined by you:

- Press the F2 (**STEP**) handwheel soft key
- To activate incremental jog positioning: Press handwheel soft key 3 (**ON**)
- Select the desired jog increment by pressing the **F1** or **F2** key. The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches).
- Confirm the selected jog increment with soft key 4 (**OK**)
- With the **+** or **-** handwheel key, move the active handwheel axis in the corresponding direction

> If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.
>
> By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Inputting miscellaneous functions M

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

Inputting spindle speed S

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F2** (**S**)
- Select the desired speed by pressing the **F1** or **F2** key
- Activate the new speed S with the **NC Start** key

> If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.
>
> By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Entering the feed rate F

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F3** (**F**)
- Select the desired feed rate by pressing the **F1** or **F2** key
- Load the new feed rate F with the **F3** (**OK**) handwheel soft key

> If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.
>
> By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Setting a preset

- Press handwheel soft key **F3** (**MSF**)
- Press the handwheel soft key **F4** (**PRS**)
- If required, select the axis in which the preset is to be set
- Zero 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 operating modes

With the **F4** (**OPM**) handwheel soft key you can use the handwheel to switch the mode of operation if 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 w/ Manual Data Input**
    SGL: **Program run, single block**
    RUN: **Program run, full sequence**

Generating a complete traversing block

> Your machine tool builder can assign any function to the **Generate NC block** handwheel key.

- Select the **Positioning w/ Manual Data Input** operating mode
- If required, use the arrow keys on the keyboard to select the NC block after which the new traversing block is to be inserted
- Activate the handwheel
- Press the **Generate NC block** key on the handwheel
- > The control inserts a complete traversing block containing all axis positions selected through the MOD function.

Functions in the Program Run Operating Modes

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

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel 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: Traverse manual axes (handwheel soft keys **MOP** and then **MAN**)
- Return to the contour after axes were manually traversed during a program interruption (handwheel soft keys **MOP** and then **REPO**). The handwheel soft keys, which function similarly to the screen soft keys, are used for operating.**Further information:** "Returning to the contour", page 462
- Switch on/off the “Tilt working plane” function (handwheel soft keys **MOP** and then **3-D**)

12.3 Spindle speed S, feed rate F and miscellaneous function M

Application

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.

Further information: "Enter miscellaneous functions M", page 336

Refer to your machine manual.

The machine tool builder defines which additional functions are available on the machine.

Entering values

Spindle speed S, miscellaneous function M

S

- Select input for spindle speed: press the **S** soft key

SPINDLE SPEED S=

- Enter **1000** (spindle speed) and apply this value with the **NC Start** key

The spindle speed with the entered speed**S** is started with a miscellaneous function **M**. Input a miscellaneous function **M** in the same way.

The control shows the current spindle speed in the status display. If the spindle speed is less than 1000, the control also shows a decimal place that has been entered.

Feed rate F

After entering a feed rate **F**, confirm your entry with the **ENT** key.

The following is valid for feed rate F:

- If you enter F=0, then the feed rate that the machine tool builder has defined as minimum feed rate is effective
- If the feed rate entered exceeds the maximum value that has been defined by the machine tool builder, then the value defined by the machine tool builder is effective
- F is not lost during a power interruption
- The control displays the feed rate.

Adjusting spindle speed and feed rate

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

The feed rate potentiometer only lowers the programmed feed rate, not the feed rate calculated by the control.

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

Feed rate limit F MAX

> Refer to your machine manual.
>
> The feed-rate limit depends on the machine.

The **F MAX** soft key enables you to reduce the feed rate speed for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you enter remains active after switch-off or switch-on.

The **F MAX** soft key is available in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Positioning w/ Manual Data Input**

Procedure

To activate the feed rate limit F MAX, proceed as follows:

- Operating mode: Press the **Positioning w/ Manual Data Input** key

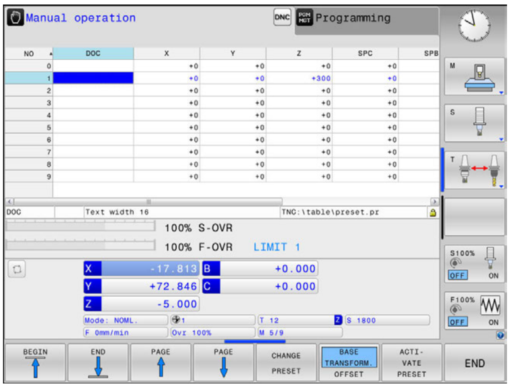
- Press the **F MAX** soft key
- Enter the desired maximum feed rate

- Press the OK soft key

12.4 Managing presets

Note

> It is essential that you use the preset table in the following cases:
> - If up to now you have been working with older controls with REF-based datum tables



The preset table can contain any number of rows (presets). To optimize the file size and the processing speed, only use as many rows as you need to manage your presets.

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

Saving presets in the table

> Refer to your machine manual.
> The machine tool builder can disable presetting in individual axes.

The preset table has the name **PRESET.PR**, and is saved in the **TNC:\table** directory. **PRESET.PR** is editable in the **Manual operation** and **Electronic handwheel** modes only if the **CHANGE PRESET** soft key was pressed. You can open the **PRESET.PR** preset table in the **Programming** operating mode but not edit it.

Copying the preset table into another directory (for data backup) is permitted. Write-protected rows are also write-protected in the copied tables.

Never change the number of rows in the copied tables! If you want to reactivate the table, this may lead to problems.

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

There are several methods for saving presets and basic rotations in the preset table:

- Manual input
- Using the probing cycles in the **Manual operation** and **Electronic handwheel** modes

> Operating notes:
> - In row 0 the control always saves the preset that you most recently set manually via the axis keys or via soft key. If the preset set manually is active, the control displays the text **PR MAN(0)** in the status display.

Manually saving the presets in the preset table

Proceed as follows in order to save presets in the preset table:

- 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 MANAGEMENT

- Press the **PRESET MANAGEMENT** soft key
- \> The control opens the preset table and sets the cursor to the row of the active preset.

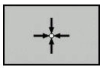
CHANGE PRESET

- Press the **CHANGE PRESET** soft key
- \> The control displays all available input options in the soft-key row.
- Select the row in the preset table that you want to change (the row number is the preset number)
- If needed, select the column in the preset table that you want to change

CORRECT THE PRESET

- Use the soft keys to select one of the available entry possibilities

Input options

Soft key	Function
	Directly transfer the actual position of the tool (the measuring dial) as the new preset: This function only saves the preset in the axis in which the cursor is currently hovering.
ENTER PRESET AGAIN	Assign any value to the actual position of the tool (the measuring dial): This function only saves the preset in the axis in which the cursor is currently hovering. Enter the desired value in the pop-up window
CORRECT THE PRESET	Incrementally shift a preset already stored in the table: This function only saves the preset in the axis in which the cursor is currently hovering. Enter the desired corrective value with the correct sign in the pop-up window. If inch display is active: Enter the value in inches, and the control will internally convert the entered values to mm
EDIT CURRENT FIELD	Directly enter the new preset without calculation of the kinematics (axis-specific). Only use this function if your machine has a rotary table, and you want to set the preset to the center of the rotary table by entering 0. This function only saves the value in the axis in which the cursor is currently hovering. Enter the desired value in the pop-up window. If inch display is active: Enter the value in inches, and the control will internally convert the entered values to mm
SAVE ACTIVE PRESET	Write the currently active preset to a selectable line in the table: This function saves the preset in all axes, and then activates the appropriate row in the table automatically. If inch display is active: Enter the value in inches, and the control will internally convert the entered values to mm

Editing the preset table

Soft key	Editing function in table mode
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
CHANGE PRESET	Select the functions for entry of presets
ACTI- VATE PRESET	Activate the preset of the selected row of the preset table
APPEND N LINES	Add multiple rows to the end of the table (2nd soft-key row)
COPY FIELD	Copy the highlighted field (2nd soft-key row)
PASTE FIELD	Insert the copied field (2nd soft-key row)
RESET LINE	Reset the selected row: The control enters – in all columns (2nd soft-key row)
INSERT LINE	Insert a single line at the end of the table (2nd soft-key row)
DELETE LINE	Delete a single line at the end of the table (2nd soft-key row)

Protecting presets from being overwritten

You can protect any rows in the preset table from being overwritten with the **LOCKED** column. The write-protected rows are color-highlighted in the preset table.

If you want to overwrite a write-protected row with a manual probing cycle, confirm with **OK** and enter the password (where password-protected).

NOTICE

Caution: Data may be lost!

Rows that were locked with the **LOCK / UNLOCK PASSWORD** function can be unlocked only with the selected password. Forgotten passwords cannot be reset. This means that locked rows would be locked permanently. The preset table would thus no longer be fully usable.

- Prefer the alternative function **LOCK / UNLOCK**
- Note down your passwords

Proceed as follows to protect a preset from being overwritten:

CHANGE PRESET

- Press the **CHANGE PRESET** soft key

→

- Select the **LOCKED** column

EDIT CURRENT FIELD

- Press the **EDIT CURRENT FIELD** soft key

Protection for a preset without using a password:

LOCK / UNLOCK

- Press the **LOCK / UNLOCK** soft key
- > The control writes an **L** in the **LOCKED** column.

Use a password to protect a preset:

LOCK / UNLOCK PASSWORD

- Press the **LOCK / UNLOCK PASSWORD** soft key
- Enter the password in the pop-up window

OK

- Confirm with the **OK** soft key or with the **ENT** key:
- > The control writes **###** in the **LOCKED** column.

Rescind write-protection

To edit a row you have previously write-protected, proceed as follows:

CHANGE PRESET

- Press the **CHANGE PRESET** soft key

→

- Select the **LOCKED** column

EDIT CURRENT FIELD

- Press the **EDIT CURRENT FIELD** soft key

Preset protected without a password:

LOCK / UNLOCK

- Press the **LOCK / UNLOCK** soft key
- > The control rescinds the write-protection.

Preset protected with a password:

LOCK / UNLOCK PASSWORD

- Press the **LOCK / UNLOCK PASSWORD** soft key
- Enter the password in the pop-up window

OK

- Confirm with the **OK** soft key or with the **ENT** key
- > The control rescinds the write-protection.

Activating a preset

Activate a preset in the Manual operation mode

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

- Before activating a preset, check whether all columns contain values.

Operating notes:

- When activating a preset from the preset table, the control resets any active datum shift, mirroring, or scaling factor.

- Select the **Manual operation** mode
- Press the **PRESET MANAGEMENT** soft key
- Select the preset number that you want to activate
- Or, with the **GOTO** key, select the preset number that you want to activate
- Confirm with the **ENT** key
- Press the **ACTIVATE PRESET** soft key
- Confirm activation of the preset
- > The control sets the display and the basic rotation.
- Exit the preset table

Activating a preset in an NC program

Use Cycle 247 in order to activate presets from the preset table during program run. In Cycle 247 you define the number of the preset to be activated.

Further information: "PRESETTING (Cycle 247)", page 601

12.5 Presetting without a 3-D touch probe

Note

When presetting, you set the control display to the coordinates of a known workpiece position.

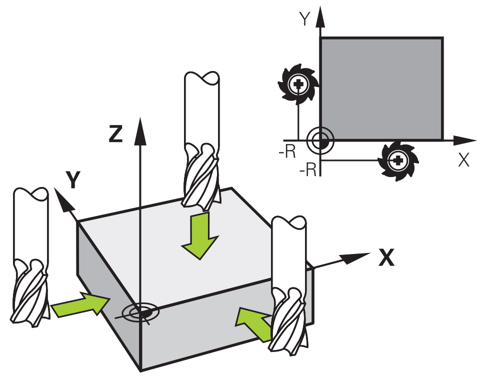
> All manual probe functions are available with a 3-D touch probe.
>
> **Further information:** "Presetting with a 3-D touch probe (option number 17)", page 418

> Refer to your machine manual.
>
> The machine tool builder can disable presetting in individual axes.

Preparation

- Clamp and align the workpiece
- Insert the zero tool with known radius into the spindle
- Ensure that the control is showing the actual positions

Presetting setting with an end mill



- Select the **Manual operation** mode

X+ Y+ Z–

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

Setting a preset in an axis:

Z

- Select the axis
- > The control opens the **PRESETTING Z=** dialog window

Alternative:

SET PRESET

- Press the **SET PRESET** soft key
- Select the axis via soft key

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 the tool in the tool axis has already been set, set the display of the tool axis to the length L of the tool or enter the sum Z=L+d.

Operating notes:

- The control automatically saves the preset set with the axis keys in row 0 of the preset table.
- If the machine tool builder has locked an axis, then you cannot set a preset in that axis. The soft key for that axis is then not visible.

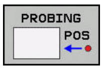
Using touch probe functions with mechanical probes or measuring dials

If you do not have an electronic 3-D touch probe on your machine, you can also use all the previously described manual touch probe functions (exception: calibration function) with mechanical probes or by simply touching the workpiece with the tool.

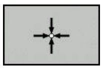
Further information: "Using a 3-D touch probe (option 17)", page 405

In place of the electronic signal generated automatically by a 3-D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key.

Proceed as follows:



- ▶ Select any touch probe function by soft key
- ▶ Move the mechanical probe to the first position to be captured by the control.



- ▶ To capture the position: Press the **Actual-position-capture** soft key
- > The control saves the current position.
- ▶ Move the mechanical probe to the next position to be captured by the control.

- ▶ To capture the position: Press the **Actual-position-capture** soft key
- > The control saves the current position.
- ▶ If required, move to additional positions and capture as described previously
- ▶ **Preset**: In the menu window, enter the coordinates of the new preset, confirm with the **SET PRESET** soft key, or write the values to a table
  Further information: "Writing measured values from the touch probe cycles to a datum table", page 410
  Further information: "Writing measured values from the touch-probe cycles to the preset table", page 411
- ▶ Terminate the probing function: Press the **END** key

If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine tool builder has defined.

12.6 Using a 3-D touch probe (option 17)

Overview

The following touch probe cycles are available in the **Manual operation** mode:

> Refer to your machine manual.
> The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.
> The touch probe cycles are available only with option number 17. If you are using a HEIDENHAIN touch probe, this option is available automatically.

> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Function	Page
CALIBRATE TS	Calibrating the 3-D Touch Probe	412
PROBING POS	Setting the preset on any axis	418
PROBING CC	Set a circle center as preset	419
PROBING CL	Setting the centerline as preset	422
TCH PROBE TABLE	Touch probe system data management	622

Traverse movements with a handwheel with display

With a handwheel with display, it is possible to transfer control to the handwheel during a manual touch probe cycle.

Proceed as follows:

- ▶ Start the manual touch probe cycle
- ▶ Position the touch probe at a position near the first touch point
- ▶ Probe the first touch point
- ▶ Activate the handwheel on the handwheel
- > The control shows the pop-up window **Handwheel active**.
- ▶ Position the touch probe at a position near the second touch point
- ▶ Deactivate the handwheel on the handwheel
- > The control closes the pop-up window.
- ▶ Probe the second touch point
- ▶ If necessary, set the preset
- ▶ End the probing function

> If the handwheel is active you cannot start the probing cycles.

Suppress touch probe monitoring

Suppress touch probe monitoring

If the stylus is deflected, the control issues an error message as soon as you want to move a machine axis.

You must deactivate touch-probe monitoring in the **Manual operation** mode in order to use a positioning block to retract a touch probe after it has deflected.

You can deactivate touch-probe monitoring for 30 seconds with the **TCH PROBE MONITOR OFF** soft key.

The control issues the error message**The touch probe monitor is deactivated for 30 seconds**. The error message automatically clears itself after 30 seconds.

> ℹ If the touch probe receives a stable signal within the 30 seconds, such as "Touch probe not deflected," then touch-probe monitoring reactivates itself automatically and the error message is cleared.

NOTICE

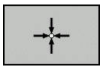
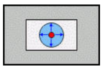
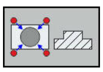
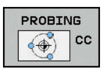
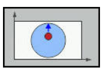
Danger of collision!

The **TCH PROBE MONITOR OFF** soft key suppresses the corresponding error message if the stylus is deflected. The control does not perform an automatic collision check with the stylus. Because of this behavior, you must check whether the touch probe can retract safely. There is a risk of collision if you choose the wrong direction for retraction.

- Carefully move the axes in the **Manual operation** mode

Functions in touch probe cycles

Soft keys that are used to select the probing direction or a probing routine are displayed in the manual touch probe cycles. The soft keys displayed vary depending on the respective cycle:

Soft key	Function
X +	Select the probing direction
	Capture the actual position
	Probe hole (inside circle) automatically
	Probe stud (outside circle) automatically
	Probe a model circle (center point of several elements)
	Select a paraxial probing direction for probing of holes, studs and model circles

Automatic probing routine for holes, studs and model circles

NOTICE
Danger of collision!
The control does not perform an automatic collision check with the stylus. During automatic probing procedures the control positions the touch probe to the probing positions automatically. There is a risk of collision if pre-positioning was not correct or if obstacles have been ignored.
▶ Program a suitable pre-position
▶ Use safety clearances to take obstacles into account

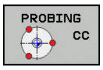
If you use a probing routine for automatic probing of a hole, stud, or a pattern circle, the control opens a form with the required entry fields.

Input fields in the Measure stud and Measure hole forms

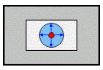
Input field	Function
Stud diameter? or **Hole diameter**	Diameter of probe contact (optional for holes)
Safety clearance?	Distance to the probe contact in the plane
Incr. clearance height?	Positioning of touch probe in spindle axis direction (starting from the current position)

Automatic probing routine:

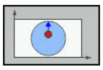
- Pre-position touch probe



- Select the probing function: Press the **PROBING CC** soft key



- Hole should be probed automatically: Press the **HOLE** soft key



- Select paraxial probing direction
- Start probing function: Press the **NC start** key
- > The control carries out all pre-positioning and probing processes automatically.

The control approaches the position at the feed rate **FMAX** defined in the touch probe table. The defined probing feed rate **F** is used for the actual probing operation.

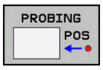
Operating and programming notes:

- Before starting an automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance opposite to the probing direction. The safety clearance is derived from the sum of the values in the touch-probe table and in the entry form.
- For inside circles with large diameters, the control can also position the touch probe on a circular arc at the feed rate **FMAX**. This requires that you enter a safety clearance for prepositioning and the hole diameter in the input form. Position the touch probe inside the hole at a position that is offset by approximately the safety clearance from the wall. Take the starting angle of the first probing process into account in pre-positioning; for example, at a starting angle of 0° the control will first probe in the positive direction of the reference axis.

Selecting the probing cycle

- ▶ Select the **Manual operation** or **Electronic handwheel** mode of operation

- ▶ Select the probing functions: Press the **TOUCH PROBE** soft key



- ▶ Select the touch probe cycle by pressing the appropriate soft key, for example **PROBING POS**
- > The control displays the associated menu.

Operating notes:

- When you select a manual probing function, the control opens a form displaying all data required. The content of the forms varies depending on the respective function.
- You can also enter values in some of the fields. Use the arrow keys to switch to the desired input field. You can position the cursor only in fields that can be edited. Fields that cannot be edited are dimmed.

Recording measured values from the touch probe cycles

Refer to your machine manual.

The control must be specially prepared by the machine tool builder for use of this function.

After executing the respective touch-probe cycle, the control writes the measured values to the TCHPRMAN.html file.

If you have not defined a path in the machine parameter **FN16DefaultPath** (no. 102202), the control will store the TCHPRMAN.html file in the **TNC:** main directory.

Operating notes:

- If you run several touch probes cycles in a row, the control stores the measured values below each other.

Writing measured values from the touch probe cycles to a datum table

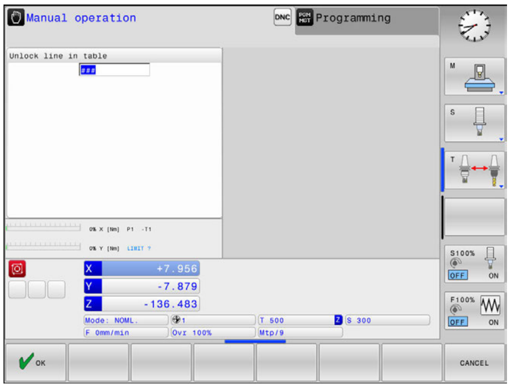
If you want to save measured values in the workpiece coordinate system, use the **ENTER IN DATUM TABLE** function. If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function.

Further information: "Writing measured values from the touch-probe cycles to the preset table", page 411

With the **ENTER IN DATUM TABLE** soft key, the control can write the values measured during any touch-probe cycle to a datum table:

- Select any probe function
- Enter the desired coordinates for the datum in the designated input boxes (depends on the touch probe cycle being run)
- Enter the datum number in the **Number in table?** input field
- Press the **ENTER IN DATUM TABLE** soft key
- > The control saves the datum in the indicated datum table under the entered number.

Writing measured values from the touch-probe cycles to the preset table



> If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function. If you want to save measured values in the workpiece coordinate system, use the **ENTER IN DATUM TABLE** function.
>
> **Further information:** "Writing measured values from the touch probe cycles to a datum table", page 410

With the **ENTRY IN PRESET TABLE** soft key, the control can write the values measured during any probe cycle in the preset table. The measured values are then stored referenced to the machine coordinate system (REF coordinates). The preset table has the name PRESET.PR, and is saved in the TNC:\table\ directory.

- ▶ Select any probe function
- ▶ Enter the desired coordinates for the preset in the designated input boxes (depends on the touch probe cycle being run)
- ▶ Enter the preset number in the **Number in table?** input field
- ▶ Press the **ENTRY IN PRESET TABLE** soft key
- > The control opens the **Overwrite active preset?** menu.
- ▶ Press the **OVERWRITE PRESET** soft key
- > The control saves the preset in the preset table under the entered number.
  - Preset number does not exist: The control saves the row only after pressing the **CREATE LINE** (Create line in table?)
  - Preset number is protected: Press the **ENTRY IN LOCKED LINE** soft key to overwrite the active preset
  - Preset number is password-protected: Press the **ENTRY IN LOCKED LINE** soft key and enter the password to overwrite the active preset

> The control displays a note if a table row cannot be written to because of disabling. The probing function itself is not interrupted.

12.7 Calibrating 3-D touch probes (option 17)

Introduction

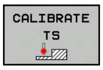
In order to precisely specify the actual trigger point of a 3-D touch probe, you must first calibrate the touch probe, otherwise the control cannot provide precise measuring results.

> Operating notes:
>
> - Always calibrate the touch probe again in the following cases:
>   - Initial configuration
>   - Broken stylus
>   - Stylus exchange
>   - Change in the probe feed rate
>   - Irregularities caused, for example, when the machine heats up
>   - Change of active tool axis
> - When you press the **OK** soft key after calibration, the calibration values are applied to the active touch probe. The updated tool data then become immediately effective, there is no need to retrieve the tool again.

During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

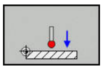
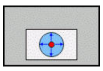
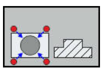
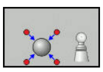
The control provides calibration cycles for calibrating the length and the radius:

- Press the **TOUCH PROBE** soft key

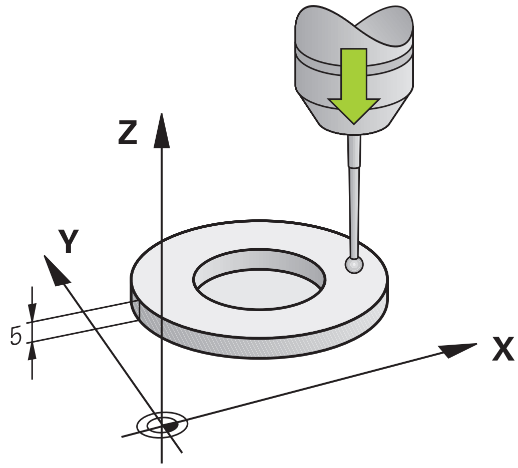


- Display the calibration cycles: Press **CALIBRATE TS**
- Select the calibration cycle

Calibration cycles

Soft key	Function	Page
	Calibrating the length	413
	Measure the radius and the center offset using a calibration ring	414
	Measure the radius and the center offset using a stud or a calibration pin	414
	Measure the radius and the center offset using a calibration sphere	414

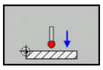
Calibrating the effective length



HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

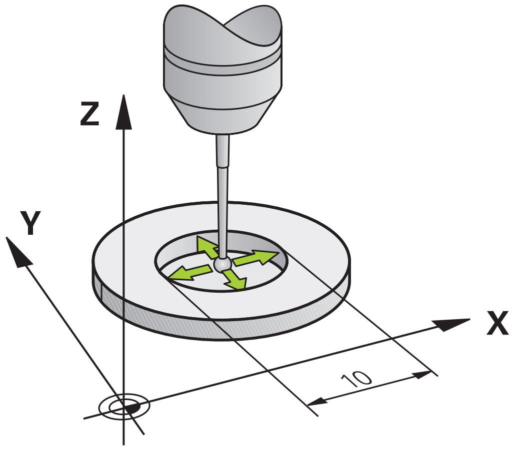
The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

- Set the preset in the spindle axis such that for the machine tool table Z=0.



- Select the calibration function for the touch probe length: Press the **CAL.** Press **L**
- > The control displays the current calibration data.
- **Datum for length?**: Enter the height of the ring gauge in the menu window
- Move the touch probe to a position just above the ring gauge
- To change the traverse direction (if necessary), press a soft key or an arrow key
- Probe surface: Press **NC Start** key
- Check results
- Press the **OK** soft key for the values to take effect
- Press the **CANCEL** soft key to terminate the calibrating function.
- > The control logs the calibration process in the TCHPRMAN.html file.

Calibrating the effective radius and compensating center misalignment



> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or stud (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

> The center offset can be determined only with a suitable touch probe.
>
> If you want to calibrate using the outside of an object, you need to preposition the touch probe above the center of the calibration sphere or calibration pin. Ensure that the touch points can be approached without collision.

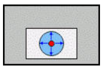
The calibration routine varies depending on how your touch probe can be oriented:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes another probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN touch probes with infrared transmission): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes another probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.

Calibration using a calibration ring

Proceed as follows for manual calibration using a calibration ring:

- In the **Manual operation** mode, position the ball tip inside the bore of the ring gauge



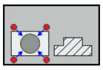
- Select the calibration function: Press the **CAL. R** soft key
- \> The control displays the current calibration data.
- Enter the diameter of the ring gauge
- Probe: Press the **NC Start** key
- \> The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results
- Press the **OK** soft key for the values to take effect
- Press the **END** soft key to terminate the calibrating function.
- \> The control logs the calibration process in the TCHPRMAN.html file.

> Refer to your machine manual.
>
> In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Calibration with a stud or calibration pin

Proceed as follows for manual calibration with a stud or calibration pin:

- In the **Manual operation** mode, position the ball tip above the center of the calibration pin



- Select the calibration function: Press the **CAL. R** soft key
- Enter the outside diameter of the stud
- Enter the safety clearance
- Probe: Press the **NC Start** key
- > The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results
- Press the **OK** soft key for the values to take effect
- Press the **END** soft key to terminate the calibrating function.
- > The control logs the calibration process in the TCHPRMAN.html file.

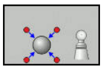
Refer to your machine manual.

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Calibration using a calibration sphere

Proceed as follows for manual calibration using a calibration sphere:

- In the **Manual operation** mode, position the ball tip above the center of the calibration sphere



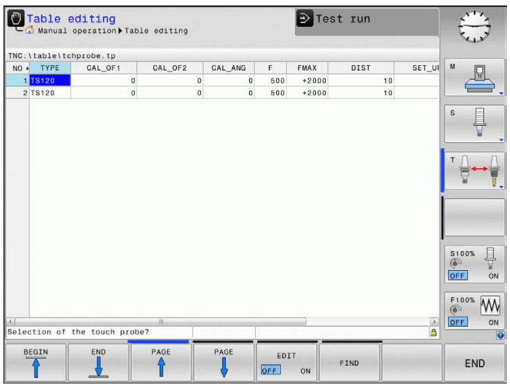
- Select the calibration function: Press the **CAL. R** soft key
- Enter the outside diameter of the ball
- Enter the safety clearance
- Select Length measurement, if applicable
- If necessary, input the reference for the length
- Probe: Press the **NC Start** key
- > The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results
- Press the **OK** soft key for the values to take effect
- Press the **END** soft key to terminate the calibrating function
- > The control logs the calibration process in the TCHPRMAN.html file.

> Refer to your machine manual.
> In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Displaying calibration values

The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (principal axis) and **CAL_OF2** (minor axis). You can display the values on the screen by pressing the **TCH PROBE TABLE** soft key.

During calibration, the control automatically creates the TCHPRMAN.html log file to which the calibration values are saved.



> Ensure that the tool number of the tool table and the touch-probe number of the touch-probe table are correct.

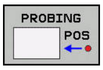
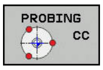
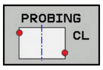
12.8 Presetting with a 3-D touch probe (option number 17)

Overview

> Refer to your machine manual.
> The machine tool builder can disable presetting in individual axes.
> If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine tool builder has defined.

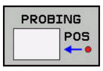
> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The following soft-key functions are available for setting a preset on an aligned workpiece:

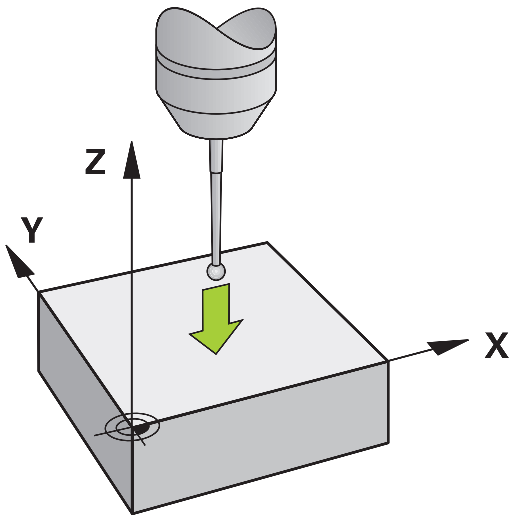
Soft key	Function	Page
	Presetting on any axis	418
	Setting a circle center as preset	419
	Center line as preset Setting the center line as preset	422

> With an active datum shift the determined value is with respect to the current preset (possibly a manual preset from the **Manual operation** mode). The datum shift is included in the position display.

Presetting on any axis



- Select the probing function by pressing the **POSITION PROBING** soft key
- Move the touch probe to a position near the touch point
- Select the axis and probing direction, e.g. Probe in direction Z-
- Probe: Press the **NC Start** key
- **Preset**: Enter the nominal coordinate
- Apply with the **SET PRESET** soft key
  Further information: "Writing measured values from the touch probe cycles to a datum table", page 410
  Further information: "Writing measured values from the touch-probe cycles to the preset table", page 411
- To terminate the probe function, press the **END** soft key



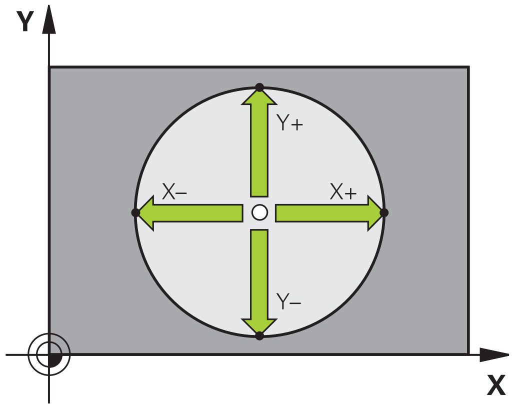
Circle center as preset

With this function, you can set the preset at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

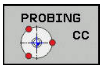
Inside circle:

The control probes the inside wall of a circle in all four coordinate axis directions.

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



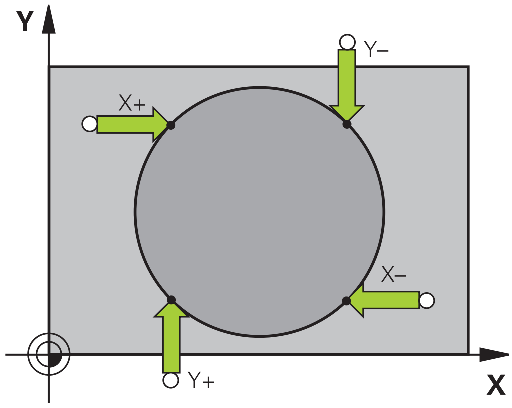
- Position the touch probe approximately in the center of the circle



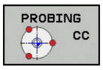
- Select the touch probe function: Press the **PROBING CC** soft key
- Select the soft key for the desired probing direction
- Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected direction. Repeat this process. After the third probing operation, you can have the control calculate the center (four touch points are recommended)
- Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- **Preset**: Enter both coordinates of the center of the circle in the menu window
- Apply with the **SET PRESET** soft key
  Further information: "Writing measured values from the touch probe cycles to a datum table", page 410
  Further information: "Writing measured values from the touch-probe cycles to the preset table", page 411
- To terminate the probe function, press the **END** soft key

> The control needs at least three touch points to calculate outside or inside circles, e.g. with circle segments. More precise results are obtained with four touch points. If possible, always pre-position the touch probe to the center.

Outside circle:



- Position the touch probe at a position near the first touch point outside of the circle



- Select the touch probe function: Press the **PROBING CC** soft key
- Select the soft key for the desired probing direction
- Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected direction. Repeat this process. After the third probing operation, you can have the control calculate the center (four touch points are recommended)
- Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- **Preset**: Enter the coordinates of the preset
- Apply with the **SET PRESET** soft key
  Further information: "Writing measured values from the touch probe cycles to a datum table", page 410
  Further information: "Writing measured values from the touch-probe cycles to the preset table", page 411
- To terminate the probe function, press the **END** soft key

Once the probing routine is completed, the control displays the current coordinates of the circle center and the circle radius.

Setting the preset using multiple holes/cylindrical studs

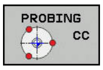
The manual probing function **Probing of circular pattern** is part of the **Cir** probing function. Individual circles can be determined with paraxial probing operations.

A second soft-key row provides the soft key **PROBING CC (Probing of circular pattern)** for using multiple holes or circular studs to set the preset. You can set the intersection of two or more elements as preset.

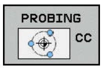
Setting the preset in the intersection of multiple holes/ circular studs:

- Pre-position touch probe

Select **Model Circle** probing function

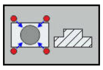


- Select the touch probe function: Press the **PROBING CC** soft key

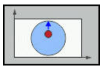


- Press the **PROBING CC (Probing of circular pattern)** soft key

Probe a circular stud



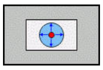
- Circular stud should be probed automatically: Press **Stud** soft key



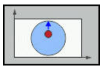
- Enter or select using soft key

- Start probing function: Press the **NC Start** key

Probe the hole.



- Hole should be probed automatically: Press the **Hole** soft key

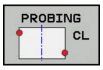


- Enter or select using soft key

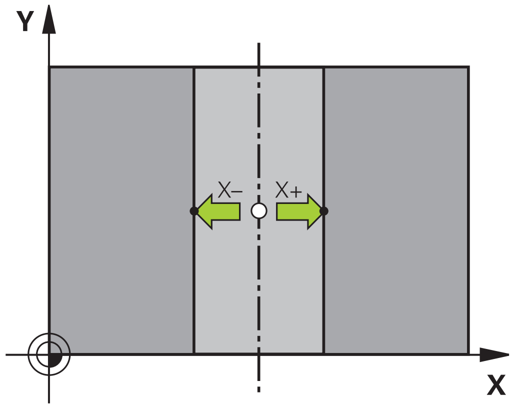
- Start probing function: Press the **NC Start** key

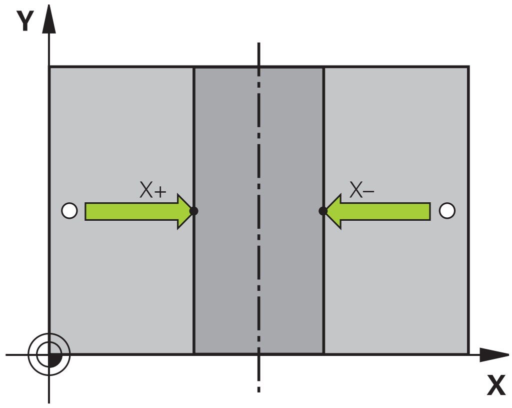
- Repeat the probing procedure for the remaining elements
- Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- **Preset**: Enter both coordinates of the center of the circle in the menu window
- Apply with the **SET PRESET** soft key
  Further information: "Writing measured values from the touch probe cycles to a datum table", page 410
  Further information: "Writing measured values from the touch-probe cycles to the preset table", page 411
- To terminate the probe function, press the **END** soft key

Setting a center line as preset



- Select the probing function: Press the **PROBING CL** soft key
- Position the touch probe at a position near the first touch point
- Select the probing direction by soft key
- Probe: Press the **NC Start** key
- Position the touch probe at a position near the second touch point
- Probe: Press the **NC Start** key
- **Preset**: Enter the coordinates of the preset in the menu window, confirm with the **SET PRESET** soft key, or write the value to a table
  Further information: "Writing measured values from the touch probe cycles to a datum table", page 410
  Further information: "Writing measured values from the touch-probe cycles to the preset table", page 411
- To terminate the probe function, press the **END** soft key





> **i** If you desire, then after the second touch point you can change the position of the centerline in the evaluation menu, and thus the axis for setting the preset. Use the soft keys to choose between principal axis, secondary axis, and tool axis. This way you can determine the positions once, and then store them in the principal axis as well as in the secondary axis.

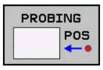
Measuring workpieces with a 3-D touch probe

You can also use the touch probe in the **Manual operation** and **Electronic handwheel** operating modes to perform simple measurements on the workpiece.

With a 3-D touch probe you can determine:

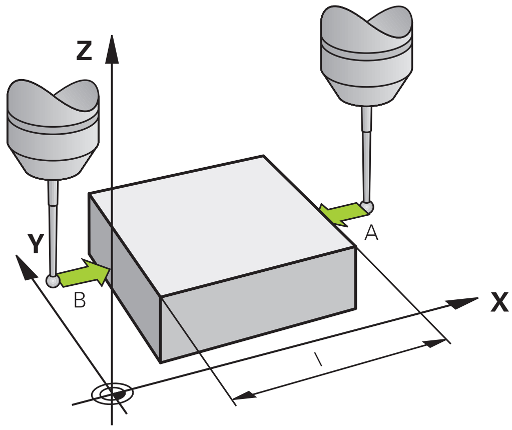
- Position coordinates, and from them,
- Dimensions on the workpiece

Finding the coordinates of a position on an aligned workpiece

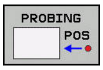


- Select the probing function: Press the **PROBING POS** soft key
- Move the touch probe to a position near the touch point
- Select the probing direction and the axis to which the coordinates relate: Use the corresponding soft keys to select
- Start the probing process: Press the **NC Start** key

The control shows the coordinates of the touch point as preset.



Measuring workpiece dimensions



- Select the probing function: Press the **PROBING POS** soft key
- Position the touch probe at a position near the first touch point A
- Select the probing direction by soft key
- Probe: Press the **NC Start** key
- If you need the current preset later, write down the value that appears in the display
- Preset: Enter **0**.
- Cancel the dialog: Press the **END** key
- Select the probing function again: Press the **PROBING POS** soft key
- Position the touch probe at a position near the second touch point B
- Select the probe direction with the soft keys: Same axis but from the opposite direction
- Probe: Press the **NC Start** key

The **Measured value** display shows the distance between the two points on the coordinate axis.

To return to the values that were active before the length measurement:

- Select the probing function: Press the **PROBING POS** soft key
- Probe the first touch point again
- Set the preset to the value that you wrote down previously
- Cancel the dialog: Press the **END** key

13

Positioning with Manual Data Input

13.1 Programming and executing simple machining operations

The **Positioning w/ Manual Data Input** mode of operation is particularly convenient for simple machining operations or for pre-positioning the tool. It enables you to write a short program in Klartext and execute it immediately. The program is stored in the file $MDI.

You can use the following functions for example:

- Cycles
- Radius compensation
- Program section repetitions
- Q parameters

The additional status display can be activated in the **Positioning w/ Manual Data Input** mode of operation.

Positioning with manual data input (MDI)

- Select the **Positioning w/ Manual Data Input** operating mode
- Program the desired available function
- Press the **NC Start** key
- > The control executes the highlighted NC block.
  Further information: "Programming and executing simple machining operations", page 426

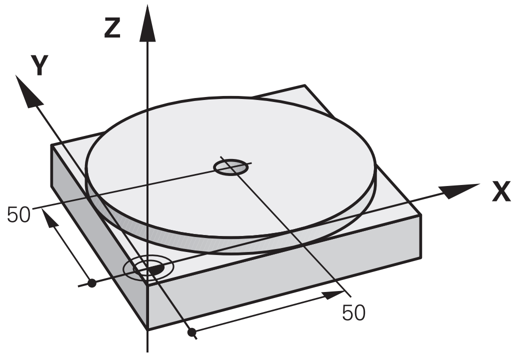
Operating and programming notes:

- The following functions are not available in the **Positioning w/ Manual Data Input** operating mode:
  - Program call
    - **PGM CALL**
    - **SEL PGM**
    - **CALL SELECTED PGM**
  - Programming graphics
  - Program-run graphics
- Using the **SELECT BLOCK** and **CUT OUT BLOCK** soft keys etc. you can also conveniently and rapidly reuse program sections from other NC programs.
  Further information: "Marking, copying, cutting and inserting program sections", page 122
- You can control and modify Q parameters with the soft keys **Q PARAMETER LIST** and **Q INFO**.
  Further information: "Checking and changing Q parameters", page 254

Example

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 preset, you can program and execute the drilling operation with a few lines of programming.

First you pre-position the tool above the workpiece with straight-line blocks and position with a safety clearance of 5 mm above the hole. Then drill the hole with Cycle **200 DRILLING**.



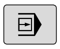
0 BEGIN PGM $MDI MM	
1 TOOL CALL 1 Z S2000	Call the tool: tool axis Z,
	spindle speed 2000 rpm
2 Z+200 R0 FMAX	Retract tool (F MAX = rapid traverse)
3 Y+50 R0 FMAX M3	Move the tool at F MAX to a position above the hole, spindle on
4 X+50 R0 FMAX	Move the tool at F MAX to a position above the hole
5 CYCL DEF 200 DRILLING	Define the DRILLING cycle
Q200=5 ;SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q201=-20 ;DEPTH	Depth of hole (algebraic sign=work direction)
Q206=250 ;FEED RATE FOR PLNGNG	Feed rate for drilling
Q202=5 ;PLUNGING DEPTH	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time after every retraction in seconds
Q203=-10 ;SURFACE COORDINATE	Coordinate of the workpiece surface
Q204=20 ;2ND SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q211=0.2 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
Q395=0 ;DEPTH REFERENCE	Depth referenced to the tool tip or the cylindrical part of the tool
6 CYCL CALL	Call the DRILLING cycle
7 Z+200 R0 FMAX M2	Retract the tool
8 END PGM $MDI MM	End of program

DRILLING cycle:

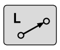
Further information: "DRILLING (Cycle 200)", page 527

Example : Remove workpiece misalignment on a machine with a rotary table

- Use a 3-D touch probe to carry out a basic rotation
  Further information: "Compensating workpiece misalignment with 3-D touch probe (option 17)", page
- Write down the rotation angle and cancel the basic rotation

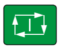


- Select the operating mode: Press the **Positioning w/ Manual Data Input** key



- Select the rotary table axis, enter the rotation angle and feed rate you wrote down, e.g. **L C +2.561 F50**

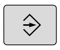
- Conclude entry



- Press the **NC Start** button: The rotation of the table corrects the misalignment

Protecting programs in $MDI

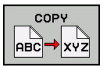
The $MDI file is intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:



- Operating mode: Press the **Programming** key

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

- Move the highlight to the **$MDI** file



- To copy the file: Press the **COPY** soft key

DESTINATION FILE =

- Enter the name under which you want to save the current contents of the $MDI file, e.g.**Hole**

- Press the **OK** soft key.

- To exit the file manager, press the **END** soft key

Further information: "Copying a single file", page 133

14

Test Run and Program Run

14.1 Graphics

Application

In the **Program run, single block** and **Program run, full sequence** operating modes, as well as in the **Test Run** Operating Mode, the control graphically simulates a machining operation.

The control features the following views:

- Plan view
- Projection in three planes
- 3-D view

> In the **Test Run** operating mode, you can additionally use the 3-D line graphics.

The graphic depicts the workpiece as if it were being machined with a cylindrical end mill.

For active tool tables, the control also takes the entries in the columns LCUTS, T-ANLGE, and R2 into consideration.

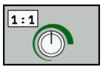
The control will not show a graphic if

- the current program has no valid workpiece blank definition
- no program is selected
- with blank form definition with a subprogram, the BLK FORM block was not yet run

Speed of the setting test runs

> **i** The most recently set speed stays active until a power interruption. After the control is switched on the speed is set to FMAX.

After you have started a program, the control displays the following soft keys with which you can set the simulation speed:

Soft key	Functions
	Test program with the speed that will be used when actually running the program (programmed feed rates will be taken into account)
	Increase the simulation speed incrementally
	Decrease the simulation speed incrementally
	Test run at the maximum possible speed (default setting)

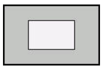
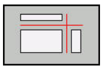
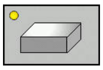
You can also set the simulation speed before you start a program:

- Select the function for setting the simulation speed

- Select the desired function by soft key, e.g. incrementally increasing the simulation speed

Overview: Display modes

In the **Program run, single block** and **Program run, full sequence** operating modes, as well as in **Test Run** operating mode, the control displays the following soft keys:

Soft key	View
	Plan view
	Projection in three planes
	3-D view

The position of the soft keys depends on the selected operating mode.

The **Test Run** mode of operation also offers the following views:

Soft key	View
	Volume view
	Volume view and tool paths
	Tool paths

Limitations during program run

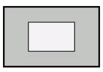
The simulation may contain errors if the control's computing capacity is being fully utilized for complex machining tasks.

Plan view

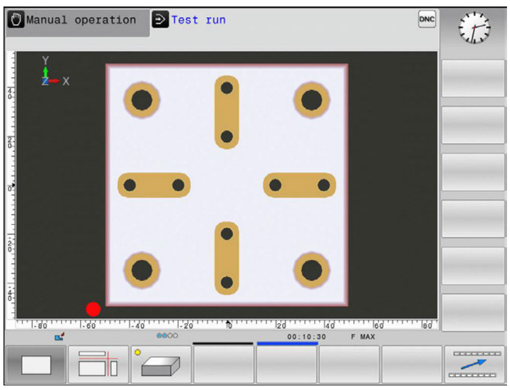
Select the plan view in the **Test Run** operating mode:

FURTHER VIEW OPTIONS

- Press the **FURTHER VIEW OPTIONS** soft key



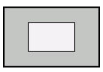
- Press the **plan view** soft key



Select plan view in the operating modes **Program run, single block** and **Program run, full sequence**:

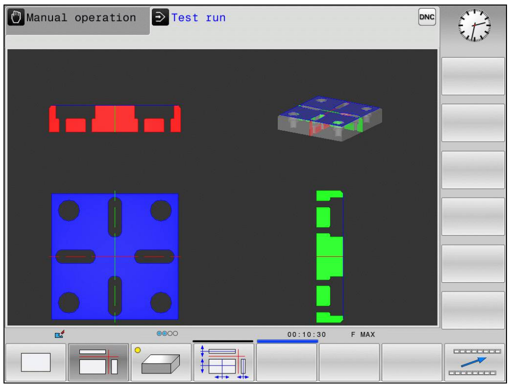
GRAPHICS

- Press the **GRAPHICS** soft key



- Press the **plan view** soft key

Projection in three planes

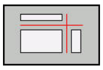


The simulation shows three sectional planes and a 3-D model, similar to a technical drawing.

Select projection in three planes in the **Test Run** operating mode:

FURTHER VIEW OPTIONS

- Press the **FURTHER VIEW OPTIONS** soft key

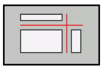


- Press the **View on 3 Planes** soft key

Select projection in three planes in the operating modes **Program run, single block** and **Program run, full sequence**:

GRAPHICS

- Press the **GRAPHICS** soft key

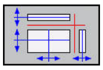


- Press the **View on 3 Planes** soft key

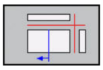
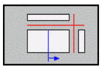
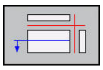
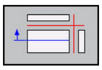
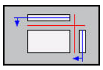
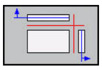
Moving sectional planes

The default setting of the sectional plane is selected so that it lies in the working plane in the workpiece center and in the tool axis on the top surface.

Shift the sectional plane as follows:



- Press the soft key for **shifting the sectional plane**
- > The control displays the following soft keys:

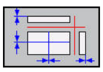
Soft keys		Function
		Shift the vertical sectional plane to the right or left
		Shift the vertical sectional plane forward or backward
		Shift the horizontal sectional plane upwards or downwards

The position of the sectional planes is visible during shifting. The shift remains active, even if you activate a new workpiece blank.

Resetting sectional planes

The shifted sectional plane also remains active for a new workpiece blank. The sectional plan is automatically reset when the control is restarted.

You can also move the sectional plane to its default position manually:

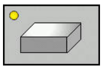


- Press the soft key for **resetting the sectional planes** soft key

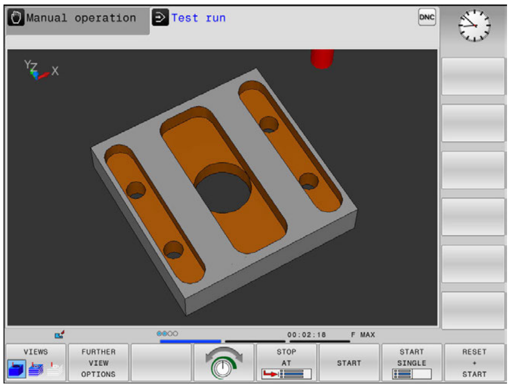
3-D view

Choose 3-D view:

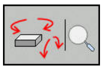
The high-resolution 3-D view enables you to display the surface of the machined workpiece in greater detail. Using a simulated light source, the control creates realistic light and shadow conditions.



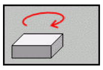
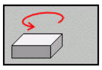
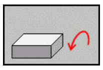
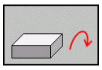
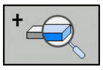
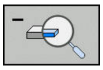
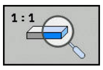
- Press the 3-D view soft key

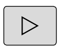


Rotating, enlarging and shifting the 3-D view

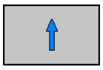
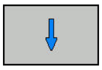
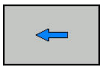
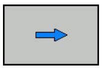
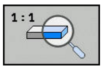


- Select functions for rotating and zooming
- > The control displays the following soft keys.

Soft keys		Function
		Rotate in 5° steps about the vertical axis
		Tilt in 5° steps about the horizontal axis
		Enlarge the graphic stepwise
		Reduce the graphic stepwise
		Reset the graphic to its original size and angle



- Scroll through the soft-key row

Soft keys		Function
		Move the graphic upward or downward
		Move the graphic to the left or right
		Reset the graphic to its original position and angle

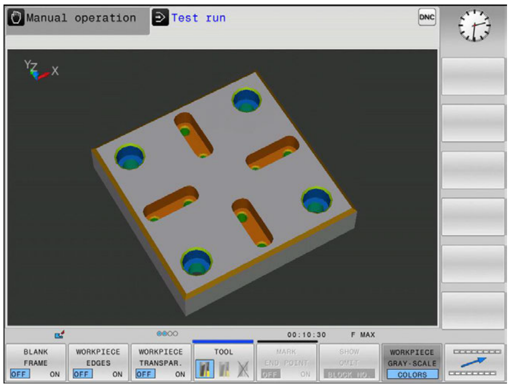
You can also use the mouse to change the graphic display. The following functions are available:

- In order to rotate the model shown in three dimensions, hold down the right mouse button and move the mouse. If you simultaneously press the shift key, you can only rotate the model horizontally or vertically
- To shift the model shown: Hold the center mouse button or mouse wheel down and move the mouse. If you simultaneously press the shift key, you can only shift the model horizontally or vertically
- To zoom in on a certain area: Mark a zoom area by holding the left mouse button down.
- > After you release the left mouse button, the control zooms in on the defined area.
- To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards
- To return to the standard display: Press the shift key and simultaneously double-click with the right mouse key. The rotation angle is maintained if you only double-click with the right mouse key

3-D view in the Test Run operating mode

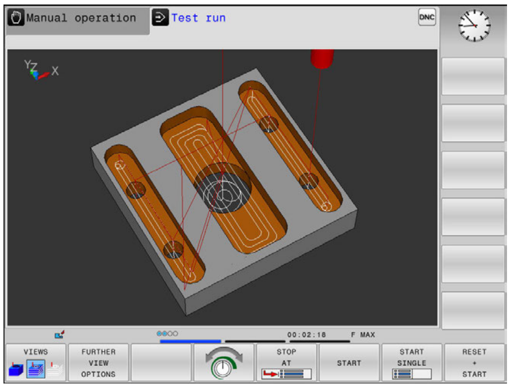
The **Test Run** mode of operation also offers the following views:

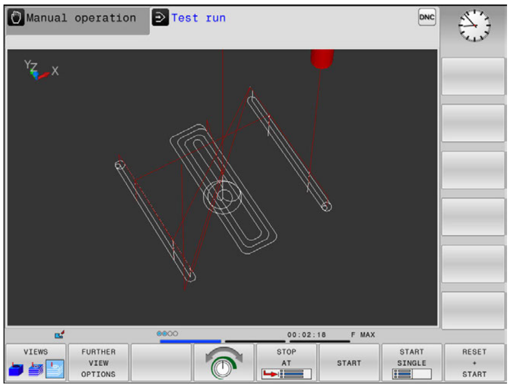
Soft keys	Function
	Volume view
	Volume view and tool paths
	Tool paths



The **Test Run** operating mode also provides the following functions:

Soft keys	Function
BLANK FRAME OFF ON	Show workpiece blank frame
WORKPIECE EDGES OFF ON	Highlight workpiece edges on 3-D model
WORKPIECE TRANSPAR. OFF ON	Show a transparent workpiece
MARK END POINT OFF ON	Show the end points of the tool paths
BLOCK NO. SHOW OMIT	Show the block numbers of the tool paths
WORKPIECE GRAY-SCALE COLORS	Show the workpiece in color
RESET THE VOLUME MODEL	Reset the volume model
RESET TOOL PATHS	Reset the tool paths
FMAX PATHS DISPLAY HIDE	Display the rapid traverse movements
MEASURING OFF ON	Activate measuring If measuring is activated, the control shows the corresponding coordinates in close proximity if you position the mouse cursor on the 3-D graphics of the workpiece.





The control saves the state of the following soft keys in non-volatile memory, even after interruption of the power supply:

- Movements at rapid traverse
- Workpiece blank frame
- Workpiece edges
- Transparent workpiece
- Workpiece in color

Operating notes:

- The available functions depend on the selected model quality. You can select the model quality in the MOD function **Graphic settings**.
- With the machine parameter **clearPathAtBlk** (No. 124203), you can specify whether or not the tool path in the **Test Run** operating mode is cleared with a new BLK FORM.
- If points were output incorrectly by the postprocessor, then machining marks occur on the workpiece. To recognize these unwanted machining marks in time (prior to machining), you can test externally created NC programs for corresponding irregularities by the display of tool paths.
- A powerful zoom function is available in order for you to quickly recognize the details for the displayed tool paths.
- The control displays traverse movements in rapid traverse in red.

Repeating graphic simulation

A part program can be graphically simulated as often as desired. To do so you can reset the graphic to the workpiece blank.

Soft key	Function
RESET BLK FORM	Display the unmachined blank in the **Program run, single block** and **Program run, full sequence** operating modes
RESET THE VOLUME MODEL	Display the unmachined blank in the **Test Run** operating mode

Tool display

You can show the tools during the simulation.

Soft key	Function
TOOLS DISPLAY HIDE	**Program run, full sequence** / **Program run, single block**
TOOL	**Test Run**

The control displays the tool in various colors:

- Red: Tool is in effect
- Blue: Tool is retracted

Measurement of machining time

Machining time in the Test Run operating mode

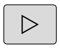
The control calculates the duration of the tool movements and displays this as machining time in the test run. The control takes feed movements and dwell times into account.

The time determined by the control is only of limited value for calculating the machining time because it does not take any machine-dependent time intervals (e.g., for tool changes) into consideration.

Machining time in the machine operating modes

Time display from program start to program end. The timer stops whenever machining is interrupted.

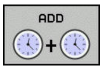
Selecting the stopwatch function



- Shift the soft key menu until the soft key for the stopwatch functions appears

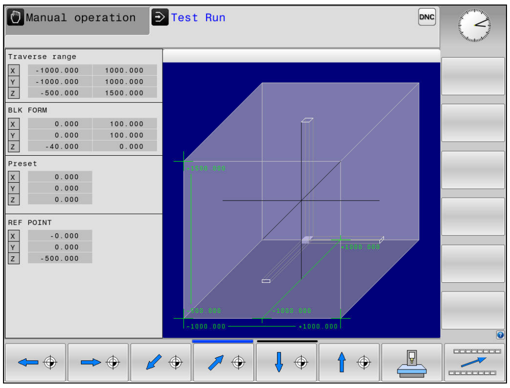
- Select the stopwatch function

- Select the desired function via soft key, e.g.,saving the displayed time

Soft key	Stopwatch functions
	Store displayed time
	Display the sum of stored time and displayed time
	Clear displayed time

14.2 Showing the workpiece blank in the working space

Application



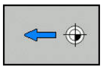
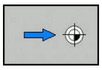
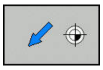
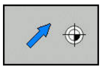
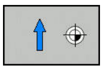
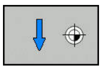
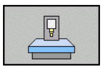
In the **Test Run** operating mode, you can graphically check the position of the workpiece blank and the preset in the working space of the machine. The graphics show the preset that has been set in the NC program using Cylce 247. If you have not set a preset in the NC program, then the graphics show the active preset on the machine.

You can active workspace monitoring in the **Test Run** operating mode: to do so, press the **BLANK IN WORK SPACE** soft key. You can activate or deactivate the function using the **SW limit monitoring** soft key.

A transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The control takes over the dimensions from the workpiece blank definition of the selected program.

For a test run it normally does not matter where the workpiece blank is located within the working space. If you activate workspace monitoring, you must graphically shift the workpiece blank so that it lies within the working space. Use the soft keys shown in the table.

You can also activate the current preset for the **Test Run** operating mode.

Soft keys		Function
		Shift workpiece blank in positive/negative X direction
		Shift workpiece blank in positive/negative Y direction
		Shift workpiece blank in positive/negative Z direction
		Show workpiece blank referenced to the set preset
ACTIVE TRAVERSE RANGES		Display the current traverse range
SELECT TRAVERSE RANGE		This shows the traverse ranges configured by the machine tool builder and can be selected accordingly.
SW limit monitoring		Switch monitoring function on or off
MACHINE REF POINT OFF ON		Display machine reference point

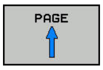
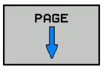
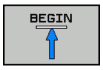
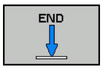
Operating notes:

- With **BLK FORM CYLINDER**, a cuboid is depicted as the workpiece blank in the working space

14.3 Functions for program display

Overview

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control displays the following soft keys for displaying the NC program in pages:

Soft key	Functions
	Go back one screen in the NC program
	Go forward one screen in the NC - program
	Select start of program
	Select end of program

14.4 Test run

Application

In the **Test Run** operating mode, you can simulate programs and program sections in order to reduce NC programming errors when programs are running. The control checks the programs for the following:

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

The following functions are also available:

- Blockwise test run
- Interruption of test at any block
- Optional block skip
- Functions for graphic simulation
- Measure machining time
- Additional status display

Keep the following in mind when performing a test run

With cuboid workpiece blanks, the control starts a test run after a tool call at the following position:

- In the working plane in the center of the defined **BLK FORM**
- In the tool axis, 1 mm above the **MAX** point defined in the **BLK FORM**

NOTICE

Danger of collision!

In the **Test Run** operating mode, the control does not take all axis movements of the machine into consideration (e.g., PLC positioning movements as well as movements from tool-change macros and M functions). This can cause a test performed without errors to later deviate from the machining operation. Danger of collision during machining!

- ▶ Test the NC program at the later machining position (**BLANK IN WORK SPACE**)
- ▶ Program a safe intermediate position after the tool change and before prepositioning
- ▶ Carefully test the NC program in the **Program run, single block** operating mode

Refer to your machine manual.

Your machine tool builder can also define a tool-change macro for the **Test Run** operating mode. This macro will simulate the exact behavior of the machine.

In doing so, the machine tool builder often changes the simulated tool change position.

Test run execution

> For the test run, you must activate a tool table (status S). Select a tool table via the file manager in the **Test Run** mode of operation.

You can select any preset table (status S) for the test run.

After **RESET + START**, line 0 of the temporarily loaded preset table automatically displays the currently active preset from **Preset.PR** (execution). Line 0 is selected when starting the test run until you define another preset in the NC program. All presets from lines > 0 are read by the control from the selected preset table of the test run.

With the **BLANK IN WORK SPACE** function, you can activate workspace monitoring for the test run.

Further information: "Showing the workpiece blank in the working space", page 442

- Operating mode: Press the **Test Run** key
- Call the file manager with the **PGM MGT** key and select the file you wish to test

The control then displays the following soft keys:

Soft key	Functions
RESET + START	Reset the blank form, reset the previous tool data and test the entire program
START	Test the entire program
START SINGLE	Test each NC block individually
STOP AT	Executes the **Test Run** until block N
STOP	Stop test run (this soft key only appears if you have started the test run)

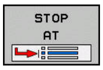
You can interrupt and continue the test run at any time, even within fixed cycles. In order to continue the test, the following actions must not be performed:

- Selecting another block with the arrow keys or the **GOTO** key
- Making changes to the program
- Selecting a new program

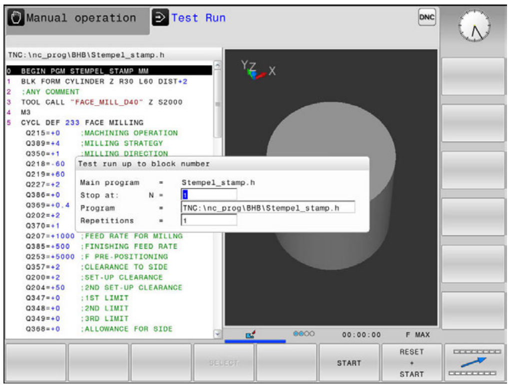
Test Run up to a certain block

With the **STOP AT** function the control executes a **Test Run** up to the block with block number **N**.

Proceed as follows to stop the **Test Run** at any block:



- Press the **STOP AT** soft key
- **Stop at: N =** Enter the block number at which the simulation should stop
- **Program =** Enter the name of the program containing the block with the selected block number
- > The control shows the name of the selected program.
- If the simulation is to be stopped in a program that has been called using **PGM CALL**, then enter this name
- **Repetitions =** If **N** is located in a program section repeat, enter the number of repeats that you want to run.
  Default 1: The control stops before **N** is simulated



Possibilities in stopped condition

If you interrupt the **Test Run** with the **STOP AT** function, you have the following possibilities in this stopped condition:

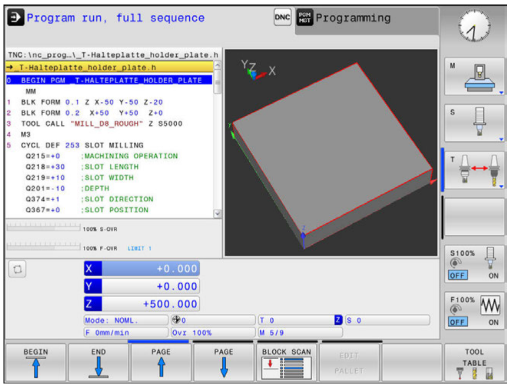
- **Block skip** enable or disable
- **Optional program stop** enable or disable
- Modify graphics resolution and model
- Modify the NC program in the **Programming** operating mode

If you modify the NC program in the **Programming** operating mode the simulation behaves as follows:

- Modification before the interruption point: The simulation restarts at the beginning
- Modification after the interruption point: Positioning at the interruption point is possible with **GOTO**

14.5 Program run

Application



In the **Program run, full sequence** operating mode, the control executes a machining program continuously to its end or up to a program stop.

In the **Program run, single block** operating mode, the control executes each block individually after pressing the **NC Start** key. With point pattern cycles and **CYCL CALL PAT** the controls stops after each point.

You can use the following control functions in the **Program run, single block** and **Program run, full sequence** operating modes:

- Interrupt program run
- Starting the program run from a certain block
- Optional block skip
- Edit the tool table TOOL.T
- Checking and changing Q parameters
- Superimpose handwheel positioning
- Functions for graphic simulation
- Additional status display

Running a part program

Preparation

1 Clamp the workpiece to the machine table.
2 Set the preset
3 Select the pallet files (status M)
4 Select the part program (status M)

Operating notes:

- You can change the feed rate and spindle speed using the potentiometers.
- You can reduce the feed rate using the **FMAX** soft key. This reduction affects all rapid traverse and feed movements, even after the control has been restarted.

Program Run, Full Sequence

- Start the machining program with the **NC Start** key

Program Run, Single Block

- Start each block of the machining program individually with the **NC Start** key

Interrupting, stopping or aborting machining

There are several ways to stop a program run:

- Interrupt the program run with e.g. the miscellaneous function **M0**
- Interrupt the program run e.g. with the miscellaneous function **M0**
- Stop the program run e.g. with the **NC stop** key in connection with the **INTERNAL STOP** soft key
- Terminate the program run e.g. with the miscellaneous functions **M2** or **M30**

The control shows the current status of the program run in the status display.

Further information: "General status display", page 85

In contrast to a stopped run, an interrupted, aborted (terminated) program run enables certain actions by the user, including the following:

- Select operating mode
- Check Q parameters and change these if necessary using the **Q INFO** function
- Change setting for the optional programmed interruption with **M1**
- Change setting for the programmed skipping of NC blocks with **/**

> During major errors, the control automatically aborts the program run (e.g., during a cycle call with stationary spindle).

Program-controlled interruptions

You can set interruptions directly in the NC program. The control interrupts the program run in the NC Block containing one of the following inputs:

- Programmed stop **M0**
- Conditional stop **M1**

NOTICE

Danger of collision!

Certain manual interactions cause the control to lose program information affecting the mode and thereby to lose the so-called contextual reference. After the loss of the contextual reference, unexpected and undesired movements can occur. There is a danger of collision during subsequent machining operations!

- Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying Q parameter values with the **Q INFO** soft key
  - Switching the operating modes
- Restore the contextual reference via repetition of the required NC blocks

Refer to your machine manual.

The miscellaneous function **M6** may also lead to a suspension of the program run. The machine manufacturer sets the functional scope of the miscellaneous functions.

Manual program interruption

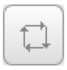
While a machining program is being executed in the **Program run, full sequence** operating mode, select the **Program run, single block** operating mode. The control interrupts the machining process at the end of the current machining step.

Abort program run.

- Press **NC Stop** key

- > The control does not exit the current NC block
- > The control shows the symbol for stopped status in the status display
- > Actions such as a change of operating mode are not possible
- > The program can be resumed with theNC Start key

- Press the **INTERNAL STOP** soft key



- > The control briefly shows the symbol for aborting the program in the status display
- > The control shows the symbol for the exited inactive status in the status display
- > Actions such as a change of operating mode are available again

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.

Modifying the preset during an interruption

If you modify the active preset during an interruption, resuming the program run is only possible with **GOTO** or mid-program startup at the interruption point.

Example:
Retracting the spindle after tool breakage

- Interrupt machining
- Enable the axis direction keys: Press the **MANUAL TRAVERSE** soft key
- Move the machine axes with the axis direction keys

> On some machines you may have to press the **NC start** key after the **MANUAL TRAVERSE** soft key to enable the axis direction keys. Refer to your machine manual.

Resuming program run after an interruption

The control saves the following data during a program interruption:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, mirroring)
- The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour after manual machine axis positioning during an interruption (**RESTORE POSITION** soft key).

Operating notes:

- The saved data remains active until it is reset (e.g., by selecting a program).
- If you interrupt an NC program using the **INTERNAL STOP** key, then you must start machining at the start of the program or by using the **BLOCK SCAN** function.
- For program interruptions within program section repeats or subprograms, re-entering at the point of interruption must be done using the **BLOCK SCAN** function.
- With machining cycles, mid-program startup is always executed at the start of the cycle. If you interrupt a program run during a machining cycle, the control repeats machining steps already carried out after a block scan.

Resuming the program run with the NC Start key

You can resume program run by pressing the machine **START** button if the program was interrupted in one of the following ways:

- Press the **NC Stop** key
- Programmed interruption

Resuming program run after an error

With an erasable error message:

- Remove the cause of the error
- Clear the error message from the screen: Press the **CE** key
- Restart the program, or resume program run where it was interrupted

Retraction after a power interruption

> Refer to your machine manual.
> Your machine tool builder configures and enables the **Retract** operating mode.

With the **Retraction** mode of operation you can disengage the tool from the workpiece after an interruption in power.

If you activated a feed rate limit before a power failure, this is still active. You can deactivate the feed-rate limit using the **CANCEL THE FEED RATE LIMITATION** soft key.

The **Retraction** mode of operation is selectable in the following conditions:

- Power interrupted
- No control voltage for the relay
- Traverse reference points

The **Retraction** operating mode offers the following modes of traverse:

Mode	Function
Machine axes	Movement of all axes in the machine coordinate system
Thread	Movements of the tool axis in the active coordinate system with compensating movement of the spindle Effective parameters: Thread pitch and direction of rotation

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are unable to reset them manually.

NOTICE

Caution: Danger to the tool and workpiece!

A power failure during the machining operation can cause uncontrolled “coasting” or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

- Use a low feed rate
- Please keep in mind that the traverse range monitoring is not available for non-referenced axes

Example

The power failed while a thread cutting cycle was being performed. You have to retract the tap:

- ▶ Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control will then display the **Power interrupted** message in the screen header.

RETRACT ON OFF

- ▶ Activate the **Retraction** mode: Press the **RETRACT** soft key
- > The control displays the message **Retraction selected**

CE

- ▶ Confirm the power interruption: Press the **CE** key
- > The control compiles the PLC program.

I

- ▶ Switch on the machine control voltage
- > The control checks the functioning of the EMERGENCY STOP circuit. If there is at least one non-referenced axis, you will have to compare the displayed position values with the actual axis values and confirm that they are correct. if required, follow the dialog.

- ▶ Check the preselected traverse mode: If required, select **THREAD**
- ▶ Check the preselected thread pitch: if required, enter the thread pitch
- ▶ Check the preselected direction of rotation: if needed, select the turning direction of the thread
  Right-handed thread: the main spindle turns clockwise when moving into the workpiece, counter-clockwise when retracting from it; left-handed thread: main spindle turns counter-clockwise when moving into the workpiece and clockwise when retracting from it

RETRACT

- ▶ Activate retraction: Press the **RETRACT** soft key

- ▶ Retraction: Retract the tool with the axis direction keys or the electronic handwheel
  Axis key Z+: Retraction from the workpiece
  Axis key Z-: Moving into the workpiece

- ▶ Exit retraction: Return to the original soft-key level

END RETRACTION

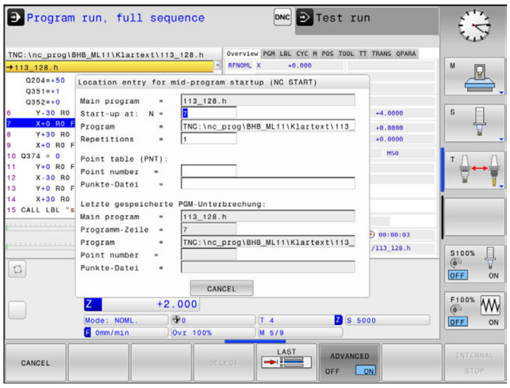
- ▶ End the **Retraction** mode: Press the **END RETRACTION** soft key
- > The control checks whether the **Retraction** mode can be ended. If necessary, follow the dialog.

- Answer confirmation request: If the tool was not correctly retracted, press the **NO** soft key. If the tool was correctly retracted, press the **YES** soft key.
- > The control hides **Retraction selected**mode.
- Initialize the machine: if required, cross the reference points
- Establish the desired machine condition: If required, reset the tilted working plane

Entering the program at any point: Mid-program startup

Refer to your machine manual.

The **BLOCK SCAN** function must be enabled and configured by the machine tool manufacturer.



With the **BLOCK SCAN** function you can start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations.

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- **INTERNAL STOP** soft key
- Emergency stop
- Power failure

If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute mid-program startup directly at the point of interruption.

You can run the mid-program startup in the following ways:

- Mid-program startup in the main program, with repetitions if necessary
- Multi-level mid-program startup in subprograms and touch probe cycles
- Mid-program startup in a point table
- Block scan in pallet programs

At the start of mid-program startup the control resets all data, as with a selection of the NC program. During the mid-program startup, you can switch between **Program Run Full Sequence** and **Program Run Single Block**.

NOTICE

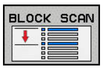
Danger of collision!

The **BLOCK SCAN** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

▶ Use the **BLOCK SCAN** function at multiple levels
Further information: "Procedure for multi-level mid-program startup", page 459

Procedure for simple mid-program startup

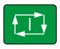
i	The control only displays the dialogs required by the process in the pop-up window.



- Press the **BLOCK SCAN** soft key
- > The control shows a pop-up window with the active main program.
- **Start-up at: N** = Enter the number of the NC block where you wish to enter the NC program
- **Program** = Check the name and path of the NC program containing the NC block, or enter with the **SELECT** soft key
- **Repetitions** = Enter the number of repetitions which should be taken into account in the block scan if the NC block is located within a program section repetition.
  Default 1 means initial machining operating

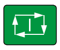
- Press the **ADVANCED** soft key if required

- if required, press the **INSERT LAST NC BLOCK** soft key to select the last saved interruption



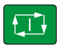
- Press the **NC Start** key
- > The control starts the block scan, calculates until the entered NC block and shows the next dialog.

If you changed the machine status:

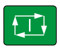


- Press the **NC Start** key
- > The control restores the machine status, e.g. TOOL CALL, M functions and shows the next dialog.

If you changed the axis positions:



- Press the **NC Start** key
- > The control approaches the specified positions in the specified sequence and shows the next dialog.
  Approach axes in individually selected sequence:
  Further information: "Returning to the contour", page 462



- Press the **NC Start** key
- > The control resumes execution of the NC program.

Example of simple mid-program startup

After an internal stop, you would like to start in block 12 in the third machining operation of LBL 1.

In the pop-up window enter the following data:

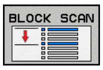
- **Start-up at: N** =12
- **Repetitions** = 3

Procedure for multi-level mid-program startup

If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level mid-program startup. For this purpose, jump in the main program to the desired subprogram call. With the **CONTINUE BLOCK SCAN** function, you can jump further from this position.

> Operating notes:
> - The control only displays the dialogs required by the process in the pop-up window.
> - You can also continue the **BLOCK SCAN** without restoring the machine status and the axis position of the first startup point. For this, press the **CONTINUE BLOCK SCAN** soft key before confirming the restoration with the **NC-Start** key.

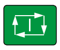
Mid-program startup to the first start-up point:



- Press the **BLOCK SCAN** soft key
- Enter the first NC block where you wish to start

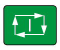
- Press the **ADVANCED** soft key if required

- If required, press the **INSERT LAST NC BLOCK** soft key in order to select the last saved interruption



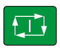
- Press the **NC Start** key
- > The control starts the block scan and calculates until the entered NC block.

If the control should restore the machine status of the entered NC block:



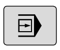
- Press the **NC Start** key
- > The control restores the machine status, e.g. TOOL CALL, M functions.

If the control should restore the axis positions:

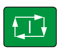


- Press the **NC Start** key
- > The control moves in the specified sequence to the specified positions.

If the control should run the NC block:

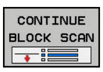


- Select the **Program Run Single Block** operating mode if required



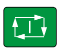
- Press the **NC Start** key
- > The control runs the NC block.

Mid-program startup to the next start-up point:

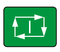


- Press the **CONTINUE BLOCK SCAN** soft key
- Enter the NC block where you wish to start

If you changed the machine status:



- Press the **NC Start** key



- Press the **NC Start** key

If the control should run the NC block:

- Press the **NC Start** key
- Repeat these steps if required to jump to the next start-up point
- Press the **NC Start** key
- > The control resumes execution of the NC program.

Example of multi-level mid-program startup

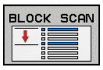
You run a main program with several subprogram calls in the program Sub.h. You work with a touch probe cycle in the main program. You use the result of the touch probe cycle later for positioning.

After an internal stop you wish to start up in block 8 in the second call of the subprogram. This subprogram call is in block 53 of the main program. The touch probe cycle is in block 28 of the main program, i.e. before the desired start-up point.

- Press the **BLOCK SCAN** soft key
- In the pop-up window enter the following data:
  - **Start-up at: N** =28
  - **Repetitions** = 1
- Select the **Program Run Single Block** operating mode if required
- Press the **NC start** key until the control runs the touch probe cycle
- > The control saves the result.
- Press the **CONTINUE BLOCK SCAN** soft key
- In the pop-up window enter the following data:
  - **Start-up at: N** =53
  - **Repetitions** = 1
- Press the **NC start** key until the control runs the NC block
- > The control jumps into the subprogram Sub.h.
- Press the **CONTINUE BLOCK SCAN** soft key
- In the pop-up window enter the following data:
  - **Start-up at: N** =8
  - **Repetitions** = 1
- Press the **NC start** key until the control runs the NC block
- > The control continues to run the subprogram and then returns to the main program.

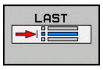
Block scan in a point table

If you start in a point table called by the main program, use the **ADVANCED** soft key.

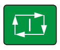


- Press the **BLOCK SCAN** soft key
- > The control shows a pop-up window.

- Press the **ADVANCED** soft key
- > The control expands the pop-up window.
- **Point number =** enter the line number of the point table you start with
- Enter the **Point file =** name and path of the point table



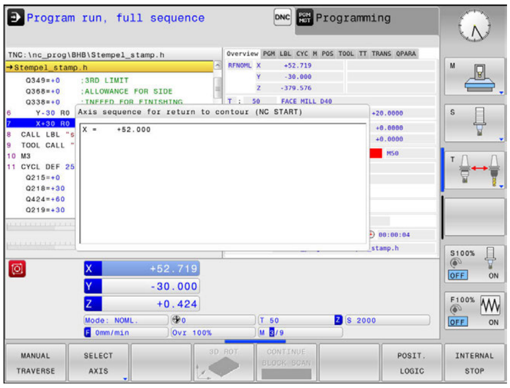
- If required, press the **SELECT LAST BLOCK** soft key in order to select the last saved interruption



- Press the **NC Start** key

If you would like to start with the mid-program startup in a point pattern, then proceed just as you would for starting in the point table. Enter the desired point number in the **Point number =** input field. The first point in the point pattern has the point number **0**.

Returning to the contour



With the **RESTORE POSITION** function, the control moves the tool 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 with 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

Procedure

Proceed as follows to approach the contour:

RESTORE POSITION

- Press the **RESTORE POSITION** soft key
- Restore the machine status, if required

Approach the axes in the sequence shown by the control:

- Press the **NC Start** key

Approach the axes according to individually selected sequence:

SELECT AXIS

- Press the **SELECT AXIS** soft key
- Press the axis soft key of the first axis
- Press the **NC Start** key
- Press the axis soft key of the second axis
- Press the **NC Start** key
- Repeat the process for all axes

> If the tool is located in the tool axis below the starting point, then the control offers the tool axis as the first traverse direction.

14.6 Skipping blocks

Application

You can have blocks skipped in the **Test Run** or **Program Run, Full Sequence/Single Block** operating modes if you have marked these blocks with a **/** sign:

- In order to not execute or not test NC blocks with a **/** sign, set the soft key to **ON**

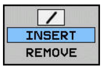
- To execute or test NC blocks with a **/** sign, set the soft key to **OFF**

Operating notes:

- This function does not work for **TOOL DEF** blocks.
- After a power interruption the control returns to the most recently selected setting.

Delete / symbol

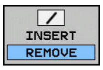
- In the **Programming** mode you select the block in which the character is to be added



- Press the **INSERT** soft key

Delete / symbol

- In the **Programming** mode you select the block in which the character is to be erased



- Press the **REMOVE** soft key

14.7 Optional program-run interruption

Application

Refer to your machine manual. The behavior of this function varies depending on the respective machine.

The control optionally interrupts program run at blocks in which an M1 has been programmed. If you use M1 in the **Program run** operating mode, then the control does not switch off the spindle or the coolant.

- Do not interrupt **Program run** or **Test Run** with blocks containing M1: Set the soft key to **OFF**

- Interrupt **Program run** or **Test Run** with blocks containing M1: Set the soft key to **ON**

15

MOD Functions

15.1 MOD function

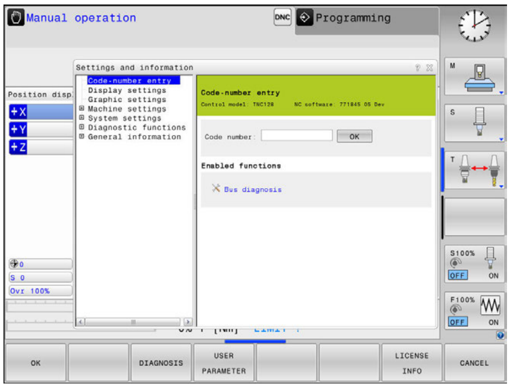
The MOD functions provide additional input possibilities and displays. In addition, you can enter code numbers to enable access to protected areas.

Selecting MOD functions

Open the pop-up window with the MOD functions:

MOD

- Press the **MOD** key
- > The control opens a pop-up window displaying the available MOD functions.



Changing the settings

There are three possibilities for changing a setting, depending on the function selected:

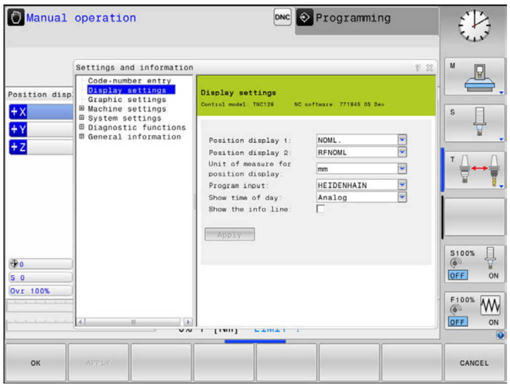
- Enter a numerical value directly, e.g., when determining the traverse range limit
- Change the setting by pressing the **ENT** key
- Change a setting via a selection window

> If there are multiple possible settings available, then you can show the selection box by pressing the **GOTO** key. Select the desired setting with the **ENT** key. If you do not wish to change the setting, close the window with the **END** key.

Exiting MOD functions

- Exit the MOD functions: Press the **CANCEL** soft key or the **END** key

Overview of MOD functions



The following functions are available independent of the selected operating mode:

Code-number entry

- Code number

Display settings

- Digital readouts
- Measuring unit (mm/inch) for position display
- Program entry for MDI
- Show time of day
- Show the info line

Graphic settings

- Model type
- Model quality

Counter settings

- Momentary count
- PGM for counter

Machine settings

- Kinematics
- Traverse limits
- Tool-usage file
- External access
- Set up wireless handwheel

System settings

- Set the system time
- Define the network connection
- Network: IP configuration

Diagnostic functions

- Bus diagnosis
- HEROS information

General information

- Version information
- License information
- Machine times

15.2 Graphic settings

With the MOD functions **Graphic settings** you can select the model type and model quality for the **Test Run** operating mode.

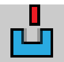
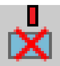
To select **Graphic settings** proceed as follows:

- Select the group **Graphic settings** from the MOD menu
- Select the model type
- Select the model quality
- Press the **APPLY** soft key
- Press the **OK** soft key.

In the **Test Run** operating mode, the control displays icons of the active **Graphic settings**.

You have the following simulation parameters for the control's **Graphic settings**:

Model type

Icon	Choice	Properties	Application
	3-D	Very true to detail, heavy time and processor consumption	Milling with undercuts,
	2.5 D	Fast	Milling without undercuts
	No model	Very fast	Line graphics

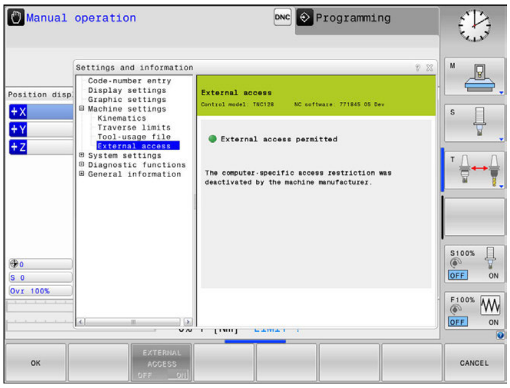
Model quality

Icon	Choice	Properties
	Very high	High data transfer rate, exact depiction of tool geometry, depiction of block end points and block numbers possible
	High	High data transfer rate, exact depiction of tool geometry
	Medium	Medium data transfer rate, approximation of tool geometry
	Low	Low data transfer rate, coarse approximation of tool geometry

15.3 Machine settings

External access

Refer to your machine manual.
The machine tool builder can configure the external access options.



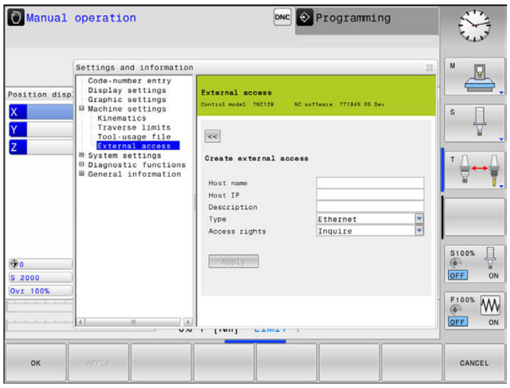
With the MOD function **External access**, you can grant or restrict access to the control. Once you have restricted external access, it is no longer possible to connect to the control and to exchange data over a network or over a serial connection (e.g., with the TNCremo data transfer software).

Proceed as follows to restrict external access:

- In the MOD menu, select the **Machine settings** group
- Select the **External access** menu
- Set the **EXTERNAL ACCESS ON/OFF** soft key to **OFF**
- Press the **OK** soft key

Computer-specific access control

If your machine manufacturer has set up computer-specific access control (machine parameter **CfgAccessControl** no. 123400)), you can permit access for up to 32 connections authorized by you. Select **Add** to create a new connection. The control then opens an input box for you to enter the connection data.



Access settings	
Host name	Host name of the external computer
Host IP	Network address of the external computer
Description	Additional information (text is shown in the overview list)
Type:	
Ethernet	Network connection
Com 1	Serial interface 1
COM 2	Serial interface 2
Access rights:	
Inquire	For external access, the control opens a query dialog
Deny	Do not permit network access
Permit	Permit network access without query

If you assign the **Inquire** access right to a connection, and if access is gained from this address, then the control opens a pop-up window. You must permit or deny external access in the pop-up window:

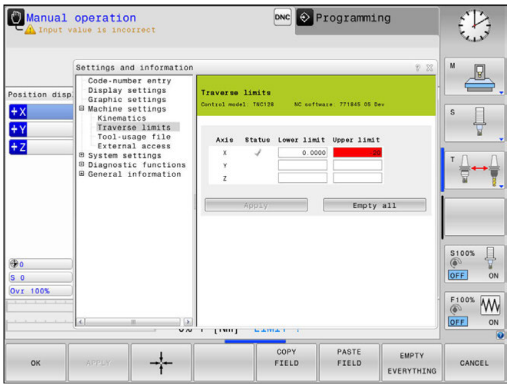
External access	Permission
Yes	Permit once
Always	Permit continuously
Never	Deny continuously
No	Deny once

> In the overview list, an active connection is shown with a green symbol.

Entering traverse limits

Refer to your machine manual.

Your machine tool builder configures and enables the**Traverse limits** function.



The MOD function **Traverse limits** enables you to limit the actually usable tool path within the maximum traverse range. This enables you to define protection zones on each axis in order, for example, to protect a component from collision.

To enter traverse limits:

- In the MOD menu, select the **Machine settings** group
- Select the **Traverse limits** menu
- Enter the values of the desired axes as a reference value or load the momentary position with the **actual position capture** soft key
- Press the **APPLY** soft key
- > The control checks the entered values for validity.
- Press the **OK** soft key

Operating notes:

- The protection zone becomes active automatically as soon as you have set a valid traverse limit in an axis. The settings are kept even after the control has been restarted.
- You can only deactivate the protection zone by deleting all values or pressing the **EMPTY EVERYTHING** soft key.

Tool usage file

Refer to your machine manual.

The tool usage test function must be enabled by your machine tool builder.

With the MOD function **Tool-usage file**, you can select whether the control never, once, or always creates a tool usage file.

Generate a tool usage file:

- In the MOD menu, select the **Machine settings** group
- Select the **Tool-usage file** menu
- Select the desired setting for the **Program Run, Full Sequence/ Single Block** and **Test Run** operating modes
- Press the **APPLY** soft key
- Press the **OK** soft key

Select kinematics

Refer to your machine manual.
Your machine tool builder configures and enables the **Kinematics selection** function.

NOTICE

Danger of collision!

All stored kinematics can also be selected as active machine kinematics. By this means, all manual movements and machining operations are executed using the selected kinematics. All subsequent axis movements pose a risk of collision!

- Use the **Kinematics selection** function only in the **Test Run** operating mode
- Use the **Kinematics selection** function for selecting the active machine kinematics only as needed

You can use this function to test programs whose kinematics does not match the active machine kinematics. If your machine manufacturer saved different kinematic configurations in your machine, you can activate one of these kinematics configurations with the MOD function. When you select a kinematics model for the test run this does not affect machine kinematics.

Ensure that you have selected the correct kinematics in the Test Run operating mode for checking your workpiece.

15.4 System settings

Set the system time

With the **Set the system time** MOD function you can set the time zone, date and time manually or with the aid of an NTP server synchronization.

Proceed as follows to set the system time:

- In the MOD menu, select the **System settings** group
- Press the **SET DATE/ TIME** soft key
- In the **Time zone** area, select the desired time zone
- Press the **NTP on** soft key in order to select the **Set the time manually** entry
- Change the date and time as needed
- Press the **OK** soft key

To set the system time with the aid of an NTP server:

- In the MOD menu, select the **System settings** group
- Press the **SET DATE/ TIME** soft key
- In the **Time zone** area, select the desired time zone
- Press the **NTP off** soft key in order to select the **Synchronize the time over NTP server** entry
- Enter hostnames or the URL of an TNP server
- Press the **Add** soft key
- Press the **OK** soft key

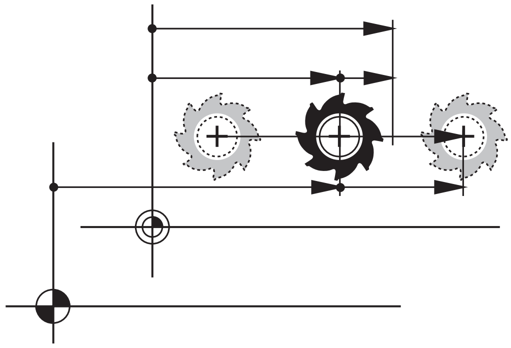
15.5 Select the position display

Application

You can influence the display of the coordinates for the operating mode **Manual operation** and the operating modes **Program run, full sequence** and **Program run, single block**.

The figure on the right shows the different tool positions:

- Initial position
- Target position of the tool
- Workpiece datum
- Machine datum



You can select the following coordinates for the control's position displays:

Display	Function
NOML	Nominal position: The value currently commanded by the control The NOML and ACTL displays differ solely with regard to following error.
ACTL	Actual position; current tool position Refer to your machine manual. Your machine tool builder defines whether the ACTL and NOML display deviates from the programmed position by the DL oversize of the tool call.
REF ACTL	Reference position; actual position relative to the machine datum
REF NOML	Reference position; nominal position relative to the machine datum
LAG	Servo lag; difference between nominal and actual positions
ACTDST	Distance remaining to the programmed position in the input coordinate system; difference between actual and target positions Examples with Cycle 11: ▶ Scaling factor 0.2 ▶ L IX+10 > The ACTDST display shows 10 mm. > The scaling factor does not have any influence.

Display	Function
REFDST	Distance remaining to the programmed position in the machine coordinate system; difference between actual and target positions Examples with Cycle 11 ▶ Scaling factor 0.2 ▶ L IX+10 > The REFDST display shows 2 mm. > The scaling factor has an effect on the distance and thus on the display.
M118	Traverse paths that were executed with handwheel superimpositioning function (**M118**)

With the MOD function **Position display 1**, you can select the position display in the status display.

With the MOD function **Position display 2**, you can select the position display in the additional status display.

15.6 Setting the unit of measure

Application

With this MOD function, you can determine whether the control coordinates are displayed in millimeters or inches.

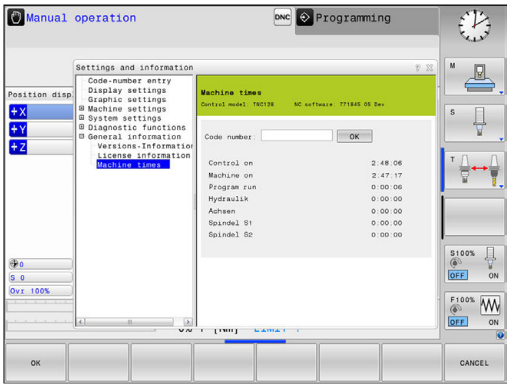
- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), value is displayed to 4 decimal places

If you would like to activate the inch display, the control shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.

15.7 Displaying operating times

Application

The **MACHINE TIME** MOD function enables you to see various types of operating times:



Operating time	Meaning
Control on	Operating time of the control since being put into service
Machine on	Operating time of the machine tool since being put into service
Program run	Duration of controlled operation since being put into service

> Refer to your machine manual.
> The machine tool builder can provide further operating time displays.

15.8 Software numbers

Application

The following software numbers are displayed on the control's screen after the **Software version** MOD function has been selected:

- **Control model**: Designation of the control (managed by HEIDENHAIN)
- **NC SW**: Number of the NC software (managed by HEIDENHAIN)
- **NCK**: Number of the NC software (managed by HEIDENHAIN)
- **PLC**: Number or name of the PLC software (managed by your machine manufacturer)

In the **FCL Information** MOD function, the control shows the following information:

- **Development level (FCL=Feature Content Level):** Development level of the software installed on the control
  Further information: "Feature Content Level (upgrade functions)", page 10

15.9 Enter the code number

Application

The control requires a code number for the following functions:

Function	Code number
Select user parameters	123
Configuring an Ethernet card	NET123
Enabling special functions for Q parameter programming	555343

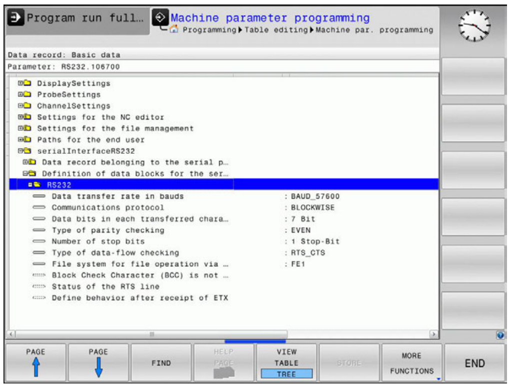
15.10 Setting up data interfaces

Serial interfaces on the TNC 128

The TNC 128 automatically uses the LSV2 transmission protocol for serial data transfer. The LSV2 protocol is prescribed and cannot be modified apart from setting the baud rate (machine parameter **baudRateLsv2,** no. 106606). You can also define another type of data transfer (interface). The settings described below are therefore effective only for the respective newly defined interface.

Application

To set up a data interface, press the **MOD** key. Enter the code number 123. In the **CfgSerialInterface** (no. 106700) machine parameter, you can enter the following settings:



Setting the RS-232 interface

Open the RS232 folder. The control then displays the following settings:

Set BAUD RATE (baud rate no. 106701)

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

Set protocol (protocol no. 106702)

The data transfer protocol controls the data flow of a serial transmission (comparable to MP5030 of the iTNC 530).

> Operating notes:
> - The **BLOCKWISE** setting designates a type of data transfer in which the data is transferred grouped in blocks.
> - The **BLOCKWISE** setting does **not** correspond to the data reception in blocks nor to the simultaneous execution of older contouring controls in blocks. This function is no longer available for current controls.

Data transmission protocol	Selection
Standard data transmission (transmission line-by-line)	STANDARD
Packet-based data transfer	BLOCKWISE
Transmission without protocol (only character-by-character)	RAW_DATA

Set data bits (dataBits no. 106703)

By setting the data bits you define whether a character is transmitted with 7 or 8 data bits.

Check parity (parity no. 106704)

The parity bit helps the receiver to detect transmission errors. The parity bit can be formed in three different ways:

- No parity (NONE): There is no error detection
- Even parity (EVEN): Here there is an error if the receiver finds that it has received an odd number of set bits
- Odd parity (ODD): Here there is an error if the receiver finds that it has received an even number of set bits

Set stop bits (stopBits no. 106705)

The start bit and one or two stop bits enable the receiver to synchronize each transmitted character during serial data transmission.

Set handshake (flowControl no. 106706)

By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking.

- No data flow checking (NONE): Handshaking is not active
- Hardware handshaking (RTS_CTS): Transmission stop is active through RTS
- Software handshaking (XON_XOFF): Transmission stop is active through DC3 (XOFF)

File system for file operation (fileSystem no. 106707)

In **fileSystem** you define the file system for the serial interface. This machine parameter is not required if you don't need a special file system.

- EXT: Minimum file system for printers or non-HEIDENHAIN transmission software. It corresponds to the EXT1 and EXT2 operating modes on older HEIDENHAIN controls.
- FE1: Communication with the TNCserver PC software or an external floppy disk unit.

Block check character (bccAvoidCtrlChar no. 106708)

With Block Check Character (optional) no control character, you determine whether the checksum can correspond to a control character.

- TRUE: The checksum does not correspond to a control character
- FALSE: The checksum can correspond to a control character

Condition of RTS line (rtsLow no. 106709)

With the state of the RTS line (optional), you can define whether the **LOW** level is active in idle state.

- TRUE: Level is **LOW** in idle state
- FALSE: Level is not **LOW** in idle state

Define behavior after receipt of ETX (noEotAfterEtx no. 106710)

With define behavior after reception of ETX (optional) you determine whether the EOT character is sent after the ETX character was received.

- TRUE: The EOT character is not sent
- FALSE: The EOT character is sent

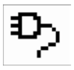
Settings for the transmission of data using PC software TNCserver

Apply the following settings in machine parameter **RS232** (no. 106700):

Parameters	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Specify type of handshake:	RTS_CTS
File system for file operations	FE1

Setting the operating mode of the external device (fileSystem)

> The **load all programs**, **load offered program**, and **load directory** functions are not available in the **FE2** and **FEX** operating modes.

Icon	External device	Operating mode
	PC with HEIDENHAIN TNCremo data transfer software	LSV2
	HEIDENHAIN floppy disk units	FE1
	Non-HEIDENHAIN devices such as printers, scanners, punchers, PC without TNCremo	FEX

Software for data transfer

For data transfer to or from the control, you should use the HEIDENHAIN TNCremo software. With TNCremo, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.

> You can download the current version of the **TNCremo** software from the HEIDENHAIN homepage.

System requirements for TNCremo:

- PC with 486 processor or higher
- Windows XP, Windows Vista, Windows 7, Windows 8 operating system
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

Installation under Windows

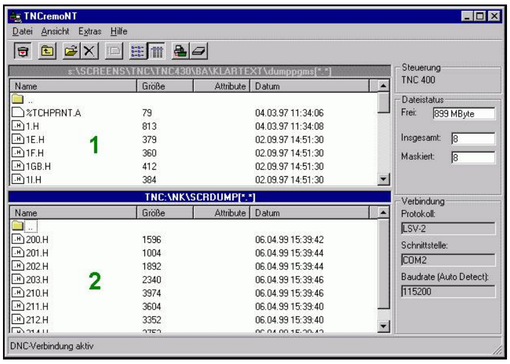
- ▶ Start the SETUP.EXE installation program with the file manager (Explorer)
- ▶ Follow the setup program instructions

Starting TNCremo under Windows

- ▶ Click on <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremo>

When you start TNCremo for the first time, it automatically tries to set up a connection with the control.

Data transfer between the control and TNCremo



Check whether the control is connected to the correct serial port on your PC or to the network.

Once you have started TNCremo, 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 <File>, <Change directory>, you can select any drive or 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>. TNCremo now receives the file and directory structure from the control and displays this in the lower part of the main window **2**
- To transfer a file from the control to the PC, select the file in the control window per mouse click and move the highlighted file into the PC window while holding down the mouse button **1**
- To transfer a file from the PC to the control, select the file in the PC window per mouse click and move the highlighted file into the control window while holding down the mouse button **2**

If you want to control data transfer from the control, establish the connection with your PC in the following manner:

- Select <Extras>, <TNCserver>. TNCremo then starts in server mode and can receive data from the control or send data to the control
- You can now call the file management functions on the control by pressing the **PGM MGT** key in order to transfer the desired files
  Further information: "Data transfer to or from an external data carrier", page 149

> If you have exported a tool table from the control, then the tool types are converted to tool type numbers.

End TNCremo

Select <File>, <Exit>

> You can open the context-sensitive help function of the **TNCremo** software by pressing the **F1** key.

15.11 Ethernet interface

Introduction

The control is shipped with a standard Ethernet card to connect the control as a client in your network. The control transmits data via the Ethernet card with

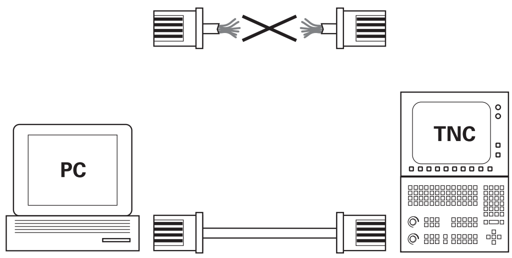
- the **smb** protocol (Server Message Block) for Windows operating systems, or
- The **TCP/IP** protocol family (**T**ransmission **C**ontrol **P**rotocol/**I**nternet **P**rotocol) and with support from the NFS (**N**etwork **F**ile **S**ystem)

> Protect your data and your control by running your machines in a secure network.

Connection possibility

You can connect the Ethernet card in your control to your network through the RJ45 connection (X26,1000BaseTX, 100BaseTX and 10BaseT), or directly to a PC. The connection is metallically isolated from the control electronics.

For a 1000Base TX, 100BaseTX, and 10BaseT connection, use a twisted-pair cable to connect the control to your network.



10BaseT / 100BaseTx / 1000BaseTx

> The maximum possible cable length depends on the quality grade of the cable, the sheathing, and the type of network (1000BaseTX, 100BaseTX, or 10BaseT)

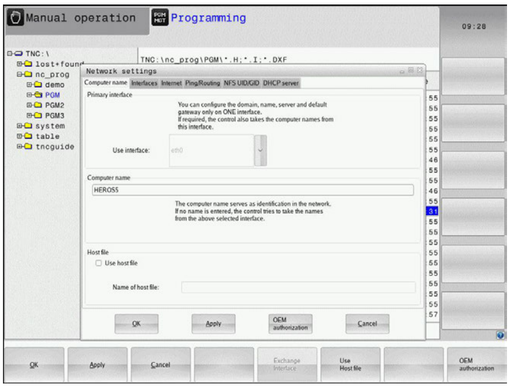
Configuring the control

> Have a network specialist configure the control.

- Press the **MOD** key
- Enter the code number **NET123**
- Press the **PGM MGT** key
- Press the **NET** soft key

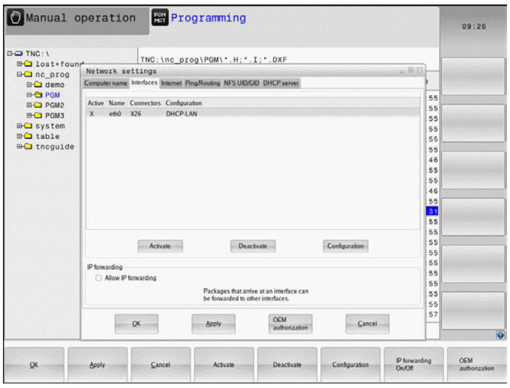
General network settings

- Press the **CONFIGURE NETWORK** soft key to enter the general network settings. The **Computer name** tab is active:



Setting	Meaning
Primary interface	Name of the Ethernet interface to be integrated in your company network. Only active if a second, optional Ethernet interface is available on the control hardware
Computer name	Name displayed for the control in your company network
Host file	**Only required for special applications**: Name of a file in which the assignments of IP addresses to computer names is defined

- Select the **Interfaces** tab to enter the interface settings:



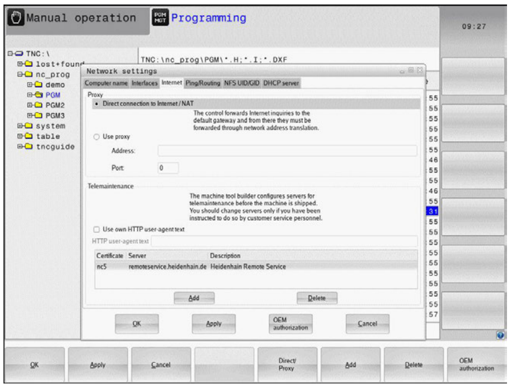
Setting	Meaning
Interface list	List of the active Ethernet interfaces. Select one of the listed interfaces (via mouse or arrow keys) ■ **Activate** button: Activate the selected interface (**X** appears in the **Active** column) ■ **Deactivate** button: Deactivate the selected interface (**-** appears in the **Active** column) ■ **Configuration** button: Open the configuration menu
Allow IP forwarding	**This function must be kept deactivated**. Only activate this function if the optionally available second Ethernet interface should be accessed externally for diagnostic purposes via the control. Only do so after instruction by our Service Department

- Press the **Configuration** button to open the Configuration menu:

Setting	Meaning
Status	■ **Interface active**: Connection status of the selected Ethernet interface ■ **Name**: Name of the interface you are currently configuring ■ **Plug connection:** Number of the plug connection of this interface on the logic unit of the control
Profile	Here you can create or select a profile in which all settings shown in this window are stored. HEIDENHAIN provides two standard profiles: ■ **DHCP-LAN**: Settings for the standard Ethernet interface; should work in a standard company network ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network Press the corresponding buttons to save, load and delete profiles
IP address	■ **Automatically procure IP address**: The control is to procure the IP address from the DHCP server ■ **Manually set IP address**: Manually define the IP address and subnet mask. Input: Four numerical values separated by periods, e.g.**160.1.180.20** and **255.255.0.0**
Domain Name Server (DNS)	■ **Automatically procure DNS**: The control is to automatically procure the IP address of the domain name server ■ **Manually configure the DNS**: Manually enter the IP addresses of the servers and the domain name
Default gateway	■ **Automatically procure default gateway**: The control is to automatically procure the default gateway ■ **Manually configure the default gateway**: Manually enter the IP addresses of the default gateway

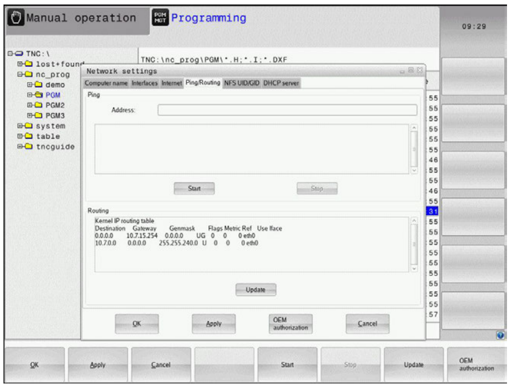
- Apply the changes with the **OK** button, or discard them with the **Cancel** button

- The **Internet** tab currently has no function.



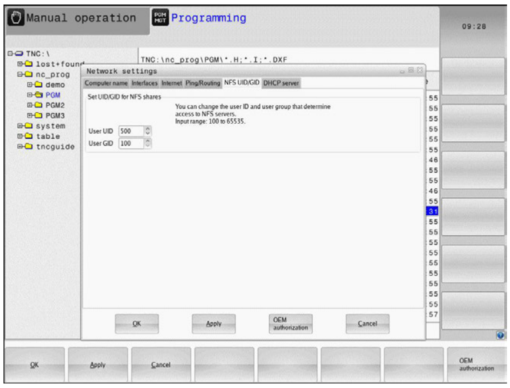
Setting	Meaning
Proxy	■ **Direct connection to Internet / NAT**: The control forwards Internet inquiries to the default gateway and from there they must be forwarded through network address translation (e.g. if a direct connection to a modem is available) ■ **Use proxy**: Define the **Address** and **Port** of the Internet router in your network, ask your network administrator for the correct address and port
Telemaintenance	The machine manufacturer configures the server for telemaintenance here. Changes must always be made in agreement with your machine tool builder

- Select the **Ping/Routing** tab to enter the ping and routing settings:



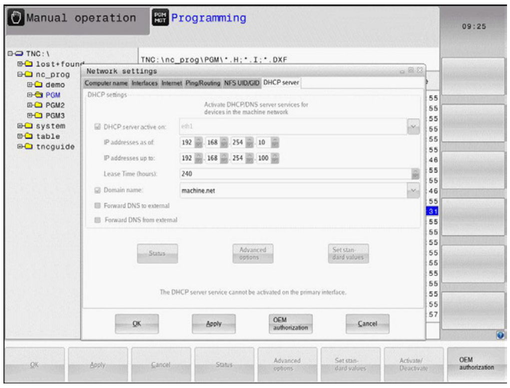
Setting	Meaning
Ping	In the **Address:** field, enter the IP number for which you want to check the network connection. Input: four numerical values separated by periods, e.g.**160.1.180.20**. As an alternative, you can enter the name of the computer whose connection you want to check ■ Press the **Start** button to begin the test. The control shows the status information in the Ping field ■ Press the **Stop** button to conclude the test
Routing	For network specialists: Status information of the operating system for the current routing ■ Press the **Update** button to refresh the routing information

- Select the **NFS UID/GID** tab to enter the user and group identifications:



Setting	Meaning
Set UID/GID for NFS shares	■ **User ID**: Definition of which user identification the end user uses to access files in the network. Ask your network specialist for the proper value ■ **Group ID**: Definition of the group identification with which you access files in the network. Ask your network specialist for the proper value

▶ **DHCP server**: Settings for automatic network configuration



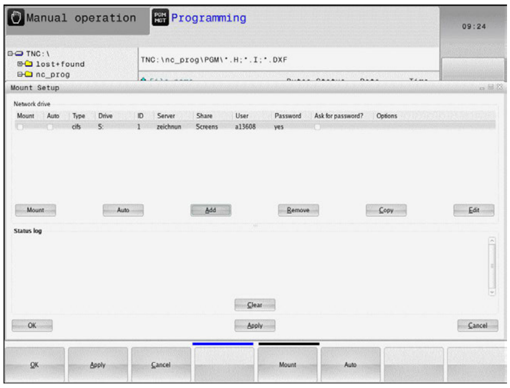
Setting	Meaning
DHCP server	■ **IP addresses from**: Define the IP address as of which the control is to derive the pool of dynamic IP addresses. The control transfers the values that appear dimmed from the static IP address of the defined Ethernet interface; these values cannot be edited. ■ **IP addresses to**: Define the IP address up to which the control is to derive the pool of dynamic IP addresses. ■ **Lease Time (hours)**: Time within which the dynamic IP address is to remain reserved for a client. If a client logs on within this time, the control reassigns the same dynamic IP address. ■ **Domain name**: Here you can define a name for the machine network if required. This is necessary if thesame names are assigned in the machine network and in the external network, for example. ■ **Forward DNS to external**: If **IP Forwarding** is active (Interfaces tab) and the option is active, you can specify that the name resolution for devices in the machine network can also be used by the external network. ■ **Forward DNS from external**: If **IP Forwarding** is active (Interfaces tab) and the option is active, you can specify that the control is to forward DNS inquiries from devices within the machine network to the name server of the external network if the DNS server of the MC cannot answer the inquiry. ■ **Status** button: Call an overview of the devices that are provided with a dynamic IP address in the machine network. You can also select settings for these devices. ■ **Advanced options** button: Additional settings for the DNS/DHCP server. ■ **Set stan- dard values** button: Set factory settings.

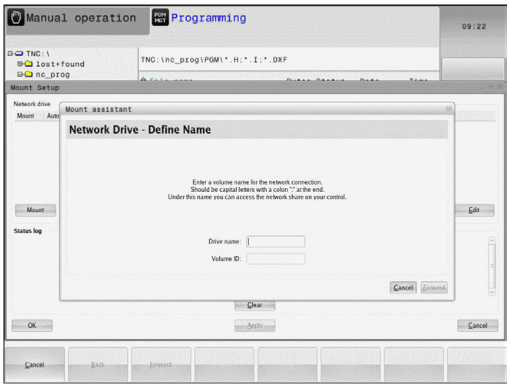
- **Sandbox**: Settings for the so-called sandbox

> Configure and use the sandbox on your control. For safety and security reasons, always open the browser in the sandbox.

Network settings specific to the device

- Press the **DEFINE NETWORK CONNECTN.** soft key 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





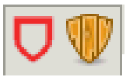
Setting	Meaning
Network drive	List of all connected network drives. The control shows the respective status of the network connections in the columns: ■ **Mount**: Network drive connected / not connected ■ **Auto**: Network drive is to be connected automatically/manually ■ **Type**: Type of network connection. cifs and nfs are possible ■ **Drive**: Designation of the drive on the control ■ **ID**: Internal ID that identifies if a mount point has been used for more than one connection ■ **Server**: Name of the server ■ **Share**: Name of the directory on the server that the control is to access ■ **User**: User name with which the user logs on to the network ■ **Password**: Network password protected or not ■ **Query password?**: Query / do not query password during connection ■ **Options**: Display additional connection options To manage the network drives, use the screen buttons. To add network drives, use the **Add** button: The control then starts the connection wizard, which guides you by dialog through the required definitions
Status log	Display of status information and error messages. Press the Clear button to delete the contents of the Status Log window.

15.12 Firewall

Application

You can set up a firewall for the primary network interface of the control. It can be configured so that incoming network traffic is blocked and/or a message is displayed, depending on the sender and the service. The firewall cannot be started for the second network interface of the control if it is active as the DHCP server.

Once the firewall has been activated, a symbol appears at the lower right in the taskbar. The symbol changes depending on the safety level that the firewall was activated with, and informs about the level of the safety settings:

Icon	Meaning
	Firewall protection does not yet exist although it has been activated according to the configuration. This can happen, for example, if PC names for which there are no equivalent IP addresses as yet were used in the configuration.
	Firewall active with medium security level
	Firewall active with high safety level. (All services except for the SSH are blocked)

> Have your network specialist check and, if necessary, change the standard settings.
>
> The settings in the additional tab **SSH settings** are in preparation for future enhancements and currently have no function.

Configuring the firewall

Make your firewall settings as follows:

- Use the mouse to open the task bar at the bottom edge of the screen
  Further information: "Window manager", page 91
- Press the green HEIDENHAIN button to open the JH menu
- Select the **Settings** menu item
- Select the **Firewall** menu item.

HEIDENHAIN recommends activating the firewall with the prepared default settings:

- Set the **Active** option to enable the firewall
- Press the **Set standard values** button to activate the default settings recommended by HEIDENHAIN.
- Exit the dialog with the **OK** button.

Firewall settings

Option	Meaning
Active	Switching the firewall on and off
Interface:	Selection of the **eth0** interface usually corresponds to X26 of the MC main computer. **eth1** corresponds to X116. You can check this in the network settings in the Interfaces tab. On main computer units with two Ethernet interfaces, the DHCP server is active by default for the second (non-primary) interface for the machine network. With this setting it is not possible to activate the firewall for **eth1** because the firewall and the DHCP server exclude themselves mutually
Report other inhibited packets:	Firewall active with high safety level. (All services except for the SSH are blocked)
Inhibit ICMP echo answer:	If this option is set, the control no longer responds to a PING request
Service	This column contains the short names of the services that are configured with this dialog. For the configuration it is not important here whether the services themselves have been started ■ **LSV2** contains the functionality for TNCremo and Teleservice, as well as the HEIDENHAIN DNC interface (ports 19000 to 19010) ■ **SMB** only refers to incoming SMB connections, i.e. if a Windows release is made on the NC. Outgoing SMB connections (i.e. if a Windows release is connected to the NC) cannot be prevented. ■ **SSH** stands for the Secure Shell protocol (port 22). As of HEROS 504, LSV2 can be executed securely tunneled via this SSH protocol ■ **VNC** protocol means access to the screen contents. If this service is blocked, the screen content can no longer be accessed, not even with the TeleService programs from HEIDENHAIN (e.g. screenshot). If this service is blocked, the VNC configuration dialog shows a warning from HEROS that VNC is disabled in the firewall.

Option	Meaning
Method	Under **Method** you can configure whether the service should not be available to anyone (**Prohibit all**), available to everyone (**Permit all**) or only available to some (Permit some). If you set **Permit some** you must also specify the computer (under Computer) that you wish to grant access to the respective service. If you do not specify any computer under **Computer**, the setting **Prohibit all** will automatically become active when the configuration is saved.
Log	If **Log** is activated, a **red** message is output if a network packet for this service has been blocked. A (blue) message is output if a network packet for this service was accepted
Computer	If the setting **Permit some** is selected under **Method**, the relevant computers can be specified here. The computers can be entered with their IP addresses or host names separated by commas. If a host name is used, the system checks upon closing or saving of the dialog whether the host name can be translated into an IP address. If this is not the case, an error message is displayed and the dialog does not terminate. If a valid host name is specified, this host name is translated into an IP address each time the control is started. If a computer that was entered with its name changes its IP address, you may have to restart the control or formally change the firewall configuration to ensure that the control uses the new IP address for a host name in the firewall.
Advanced options	These settings are only intended for your network specialists
Set standard values	Resets the settings to the default values recommended by HEIDENHAIN

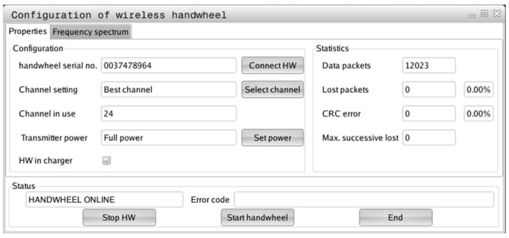
15.13 Configuring the HR 550FS wireless handwheel

Application

Press the **SET UP WIRELESS HANDWHEEL** soft key to configure the HR 550FS wireless handwheel. The following functions are available:

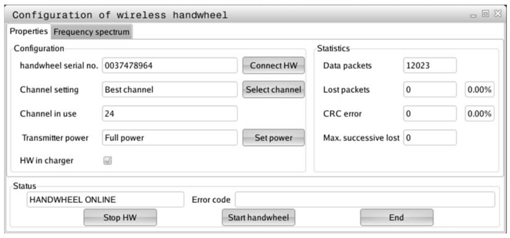
- Assigning the handwheel to a specific handwheel holder
- Setting the transmission channel
- Analyzing the frequency spectrum for determining the optimum transmission channel
- Select transmitter power
- Statistical information on the transmission quality

Assigning the handwheel to a specific handwheel holder



- ▶ Make sure that the handwheel holder is connected to the control hardware.
- ▶ Place the wireless handwheel you want to assign to the handwheel holder in the handwheel holder
- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the wireless handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click on the **Connect HW** button
- > The control saves serial number of the inserted wireless handwheel and shows it in the configuration window on the left next to the **Connect HW** button.
- ▶ To save the configuration and exit the configuration menu, press the **END** button
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the wireless handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Connect HR** button: The control saves the serial number of the wireless handwheel located in the handwheel holder and shows it in the configuration window to the left of the **Connect HR** button
- ▶ To save the configuration and exit the configuration menu, press the **END** button

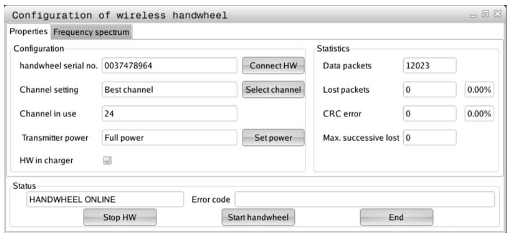
Setting the transmission channel



If the wireless handwheel is started automatically, then the control tries to select the transmission channel providing the best transmission signal. Proceed as follows if you want to set the radio channel yourself:

- Press the **MOD** key to select the MOD function
- Select the **Machine settings** menu
- Select the configuration menu for the wireless handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- Click the **Frequency spectrum** tab
- Click on the **Stop HW** button
- > The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.
- Memorize the number of the channel with the least amount of radio traffic (smallest bar)
- Click the **Start handwheel** button to reactivate the wireless handwheel
- Click the **Properties** tab
- Click on the **Select channel** button
- > The controls shows all available channel numbers
- Click the number of the channel that the control has found to have the least amount of radio traffic
- To save the configuration and exit the configuration menu, press the **END** button

Selecting the transmitter power



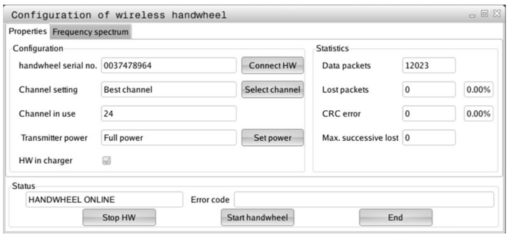
> A reduction in transmission power decreases the range of the wireless handwheel.

- Press the **MOD** key to select the MOD function
- Select the **Machine settings** menu
- Select the configuration menu for the wireless handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- Click on the **Set power** button
- > The control displays the three available power settings. Click on the desired setting.
- To save the configuration and exit the configuration menu, press the **END** button

Statistical data

To display the statistical data, proceed as follows:

- Press the **MOD** key to select the MOD function
- Select the **Machine settings** menu
- Select the configuration menu for the wireless handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- > The control displays the configuration menu with the statistical data.



Under **Statistics**, the control displays information about the transmission quality.

If the reception quality is poor so that a proper and safe stop of the axes cannot be ensured anymore, an emergency-stop reaction of the wireless handwheel is triggered.

The displayed value **Max. successive lost** indicates whether reception quality is poor. If the control repeatedly displays values greater than 2 during normal operation of the wireless handwheel within the desired range of use, then there is a high risk of an undesired disconnection. This can be corrected by increasing the transmitter power or by changing to another channel with less radio traffic.

If this occurs, try to improve the transmission quality by selecting another channel or by increasing the transmitter power.

Further information: "Setting the transmission channel", page 495
Further information: "Selecting the transmitter power", page 495

15.14 Load machine configuration

Application

NOTICE

Caution: Data may be lost!

The **RESTORE** function irrevocably overwrites the current machine configuration with the backup files. The control does not perform an automatic backup before the **RESTORE** function. The files are thus permanently gone.

- Perform a backup of the current machine configuration prior to the **RESTORE** function
- Use the function only in consultation with the machine tool builder

Your machine tool builder can provide you a backup with a machine configuration. After entering the keyword **RESTORE**, you can load the backup on your machine or programming station. Proceed as follows to load the backup:

- Enter the keyword **RESTORE** in the MOD dialog
- Select the backup file in the control's file manager (e.g., BKUP-2013-12-12_.zip)
- > The control opens the pop-up window for the backup.
- Press Emergency Stop
- Press the **OK** soft key to start the backup process

16

Fundamentals / Overviews

16.1 Introduction

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and several special functions are also available as cycles. Most cycles use Q parameters as transfer parameters.

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

- You should run a program test before machining

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

If you define a feed-rate parameter for fixed cycles greater than 200, then instead of entering a numerical value you can use soft keys to assign the feed rate defined in the **TOOL CALL** block (**FAUTO** soft key). You can also use the feed-rate alternatives **FMAX** (rapid traverse), **FZ** (feed per tooth), and **FU** (feed per rev), depending on the respective cycle and the function of the feed-rate parameter.

Note that, after a cycle definition, a change of the **FAUTO** feed rate has no effect, because internally the TNC assigns the feed rate from the **TOOL CALL** block when processing the cycle definition.

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

16.2 Available Cycle Groups

Overview of fixed cycles

CYCL DEF

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

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring, tapping and counterboring	524
POCKETS/ STUDS/ SLOTS	Cycles for milling rectangular pockets and rectangular studs	570
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	594
PATTERN	Cycles for producing point patterns	515
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop	610

▷

- If required, switch to machine-specific fixed cycles. These fixed cycles can be integrated by your machine tool builder.

16.3 Working with fixed cycles

Machine-specific cycles

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

- Cycles 300 to 399
  Machine-specific cycles that are to be defined through the **CYCLE DEF** key
- Cycles 500 to 599
  Machine-specific touch probe cycles that are to be defined through the **CYCL DEF** key

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

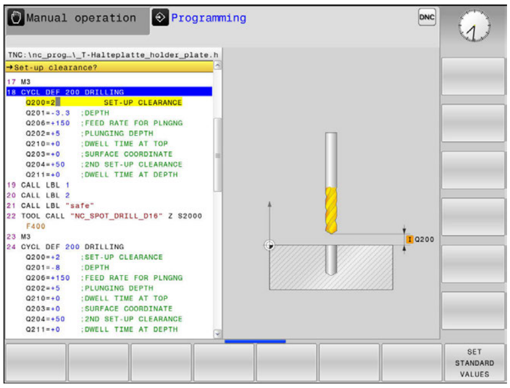
Sometimes machine-specific cycles use transfer parameters that HEIDENHAIN already uses in standard cycles. For parallel use of DEF active cycles (cycles that the TNC is automatically running during cycle definition) and CALL active cycles (cycles that you need to call up to run).

Further information: "Calling a cycle", page 504

Adhere to the following procedure in order to avoid problems regarding the overwriting of transfer parameters that are used more than once:

- 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

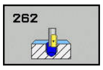


CYCL DEF

- 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



- Select the 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.
- Enter all parameters requested by the TNC and conclude each entry with the **ENT** key
- The TNC ends the dialog when all required data has been entered

Defining a cycle using the GOTO function

CYCL DEF

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

GOTO

- The TNC shows an overview of cycles in a pop-up window
- Choose the desired cycle with the arrow keys, or
- Enter the cycle number and confirm it with the **ENT** key. The TNC then initiates the cycle dialog as described above

Example NC blocks

7 CYCL DEF 200 DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=3	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
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
Q395=0	;DEPTH REFERENCE

Calling a cycle

Requirements

The following data must always be programmed before a cycle call:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Cycle definition (CYCL DEF)

For some cycles, additional prerequisites must be observed. They are detailed in the descriptions for each cycle.

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

- Cycle 220 for point patterns on circles and Cycle 221 for point patterns on lines
- Coordinate transformation cycles
- Cycle 9 DWELL TIME
- All touch probe cycles

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

Calling a cycle with CYCL CALL

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

CYCL CALL

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

Calling a cycle with CYCL CALL PAT

The **CYCL CALL PAT** function calls the most recently defined machining cycle at all positions that you defined in a PATTERN DEF pattern definition or in a points table.

Further information: "PATTERN DEF pattern definition", page 509

Further information: "Point tables", page 519

Cycle call with M99/M89

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.

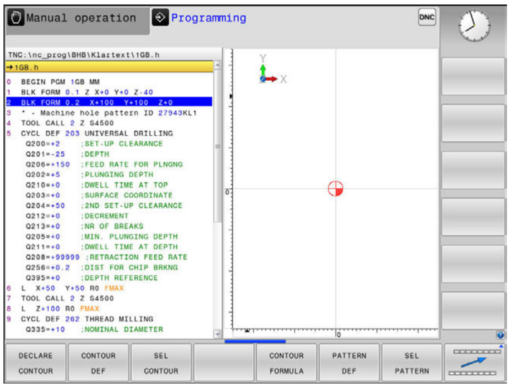
If the TNC is to run the cycle automatically after every positioning block, program the first cycle call with **M89**.

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

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

16.4 Program defaults for cycles

Overview

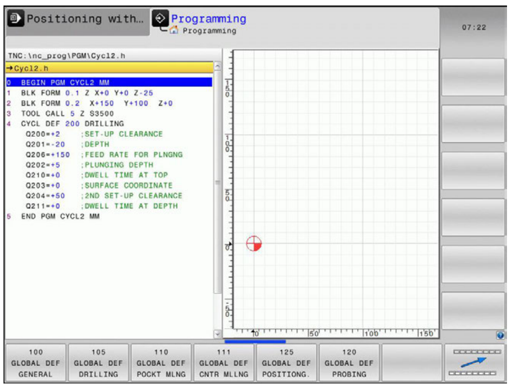


All Cycles 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200,** which you must enter for each cycle definition. The **GLOBAL DEF** function gives you the possibility of defining these cycle parameters at the beginning of the program, so that they are effective globally for all machining cycles used in the program. In the respective machining cycle you then simply link to the value defined at the beginning of the program.

The following GLOBAL DEF functions are available:

Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	507
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	508
110 GLOBAL DEF POCKT MLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	508
111 GLOBAL DEF CNTR MLLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	508
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	508
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	508

Entering GLOBAL DEF



- Mode of operation: Press the **Programming** key

SPEC FCT

- Press the SPEC FCT key to select the special functions

PROGRAM DEFAULTS

- Select the functions for program defaults

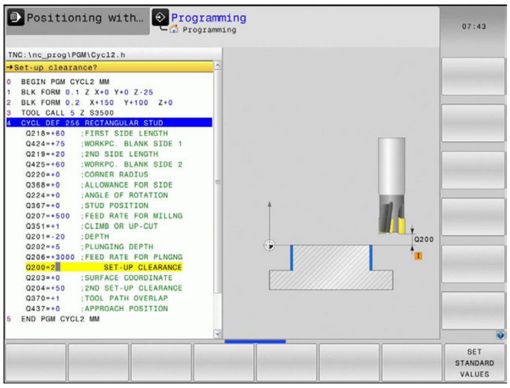
GLOBAL DEF

- Press the **GLOBAL DEF** soft key

100 GLOBAL DEF GENERAL

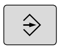
- Select the desired GLOBAL DEF function, e.g. by pressing the **GLOBAL DEF GENERAL** soft key
- Enter the required definitions, and confirm each entry with the **ENT** key

Using GLOBAL DEF information



If you entered the respective GLOBAL DEF functions at the start of the program, you can reference these globally valid values when defining any machining cycle.

Proceed as follows:



- Operating mode: Press the Programming key

- Select machining cycles: Press the CYCLE DEF key

- Select the desired cycle group, e.g. drilling cycles

- Select the desired cycle, e.g. **drilling**
- If there is a global parameter for this the TNC displays the **SET STANDARD VALUES** soft key

- Press the **SET STANDARD VALUES** soft key: The TNC inputs the word **PREDEF** in the cycle definition. You have now created a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program

NOTICE

Danger of collision!

If you modify the program settings later with **GLOBAL DEF** the modifications are effective on the complete machining program. As a consequence the machining sequence can significantly differ.

- Use **GLOBAL DEF** intentionally and run a program test before machining
- If a fixed value is entered in machining cycles, **GLOBAL DEF** does not modify this value

Global data valid everywhere

- **Safety clearance**: Distance between tool tip and workpiece surface for automated approach of the cycle start position in the tool axis
- **2nd set-up clearance:** Position to which the TNC positions the tool at the end of a machining step. The next machining position is approached at this height in the machining plane
- **F positioning:** Feed rate at which the TNC traverses the tool within a cycle
- **F retraction:** Feed rate at which the TNC retracts the tool.

The parameters are valid for all fixed cycles with numbers greater than 2xx.

Global data for drilling operations

- **Retraction rate for chip breaking:** Value by which the TNC retracts the tool during chip breaking
- **Dwell time at depth**: Time in seconds that the tool remains at the hole bottom
- **Dwell time at top**: Time in seconds that the tool remains at the set-up clearance

> The parameters apply to the drilling, tapping and thread milling cycles 200 to 209, 240 and 241.

Global data for milling operations with pocket cycles 25x

- **Overlap factor**: The tool radius multiplied by the overlap factor equals the lateral stepover
- **Climb or up-cut**: Select the type of milling
- **Plunging type**: Plunge into the material helically, in a reciprocating motion, or vertically

> The parameters apply to milling cycles 251 to 257.

Global data for milling operations with contour cycles

> Soft key **GLOBAL DEF CNTR MLLNG** has no function in the straight cut control of TNC 128. This was added for reasons of compatibility.

Global data for positioning behavior

- **Positioning behavior**: Retraction in the tool axis at the end of the machining step: Return to the 2nd set-up clearance or to the position at the beginning of the unit

> The parameters apply to each fixed cycle that you call with the **CYCL CALL PAT** function.

Global data for probing functions

- **Set-up clearance**: Distance between stylus and workpiece surface for automated approach of the probing position
- **Clearance height:** The coordinate in the touch probe axis to which the TNC traverses the touch probe between measuring points, if the **Move to clearance height** option is activated
- **Move to clearance height**: Select whether the TNC moves the touch probe to the set-up clearance or clearance height between the measuring points

> The parameters apply to all touch probe cycles numbered 4xx.

16.5 PATTERN DEF pattern definition

Application

You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. As with the cycle definitions, support graphics that illustrate the respective input parameter are also available for pattern definitions.

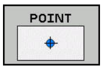
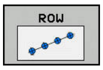
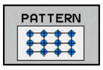
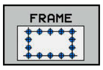
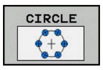
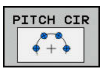
NOTICE

Danger of collision!

The **PATTERN DEF** function calculates the machining coordinates in the **X** and **Y** axes For all tools axes apart from **Z** there is a danger of collision in the following operation!

- Use **PATTERN DEF** only in connection with the tool axis **Z**

The following machining patterns are available:

Soft key	Machining pattern	Page
	POINT Definition of up to any 9 machining positions	511
	ROW Definition of a single row, straight or rotated	511
	PATTERN Definition of a single pattern, straight, rotated or distorted	512
	FRAME Definition of a single frame, straight, rotated or distorted	513
	CIRCLE Definition of a full circle	514
	PITCH CIRCLE Definition of a pitch circle	514

Entering PATTERN DEF

- Mode of operation: Press the **Programming** key
- Press the SPEC FCT key to select the special functions
- Select the functions for contour and point machining
- Press the **PATTERN DEF** soft key
- Select the desired machining pattern, e.g. press the "single row" soft key
- Enter the required definitions, and confirm each entry with the **ENT** key

Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function.

Further information: "Calling a cycle", page 504

The TNC then performs the most recently defined machining cycle on the machining pattern you defined.

> A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.
>
> The TNC retracts the tool to the clearance height between the starting points. Depending on which is greater, the TNC uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.
>
> Before **CYCL CALL PAT** you can use the function **GLOBAL DEF 125** (located in **SPEC FCT**/program defaults) with Q352=1. Then the TNC always retracts the tool between the holes to the 2nd set-up clearance that was defined in the cycle.

Defining individual machining positions

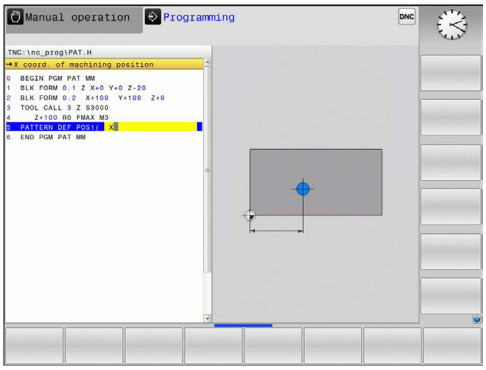
You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.

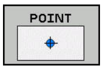
POS1 must be programmed with absolute coordinates. POS2 to POS9 can be programmed as absolute and/or incremental values.

If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

NC blocks

10 Z+100 R0 FMAX
11 PATTERN DEF **POS1 (X+25 Y+33.5 Z+0)** **POS2 (X+15 IY+6.5 Z+0)**





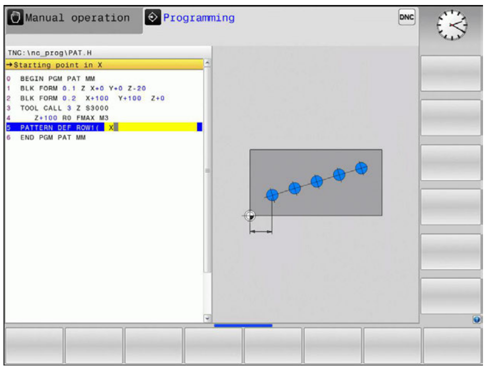
- POS1: **X coord. of machining position** (absolute): Enter X coordinate
- POS1: **Y coord. of machining position** (absolute): Enter Y coordinate
- POS1: **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin
- POS2: **X coord. of machining position** (absolute or incremental): Enter X coordinate
- POS2: **Y coord. of machining position** (absolute or incremental): Enter Y coordinate
- POS2: **Coordinate of workpiece surface** (absolute or incremental): Enter Z coordinate

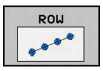
Defining a single row

If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

NC blocks

10 Z+100 R0 FMAX
11 PATTERN DEF ROW1 **(X+25 Y+33.5 D+8 NUM5 ROT+0 Z +0)**

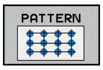




- **Starting point in X** (absolute): Coordinate of the starting point of the row in the X axis
- **Starting point in Y**(absolute): Coordinate of the starting point of the row in the Y axis
- **Spacing of machining positions** (incremental): Spacing of machining positions. You can enter a positive or negative value
- **Number of operations**: Total number of machining positions
- **Rot. position of entire pattern** (absolute): Angle of rotation around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Defining a single pattern

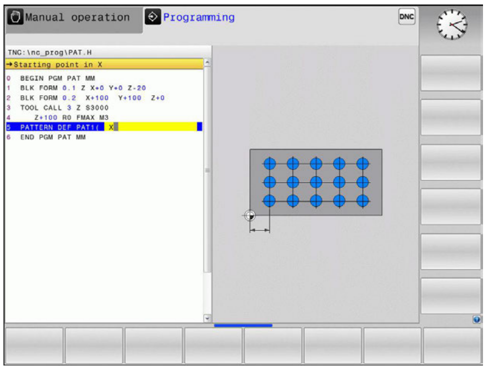
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.
>
> The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.



- **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- **Spacing of machining positions X** (incremental): Distance in X direction between the machining positions. You can enter a positive or negative value
- **Spacing of machining positions Y** (incremental): Distance in Y direction between the machining positions. You can enter a positive or negative value
- **Number of columns**: Total number of columns in the pattern
- **Number of rows**: Total number of rows in the pattern
- **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Rotary pos. minor ax.**: Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

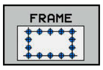
NC blocks

```
10 Z+100 R0  FMAX
11 PATTERN DEF  PAT1 (X+25 Y+33,5
    DX+8 DY+10  NUMX5 NUMY4 ROT+0
    ROTX+0 ROTY+0  Z+0)
```



Defining individual frames

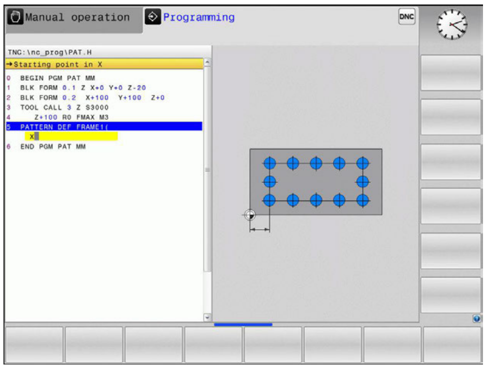
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.
>
> The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.



- **Starting point in X** (absolute): Coordinate of the starting point of the frame in the X axis
- **Starting point in Y**(absolute): Coordinate of the starting point of the frame in the Y axis
- **Spacing of machining positions X** (incremental): Distance in X direction between the machining positions. You can enter a positive or negative value
- **Spacing of machining positions Y** (incremental): Distance in Y direction between the machining positions. You can enter a positive or negative value
- **Number of columns**: Total number of columns in the pattern
- **Number of rows**: Total number of rows in the pattern
- **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Rotary pos. minor ax.**: Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

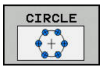
NC blocks

```
10 Z+100 R0  FMAX
11 PATTERN DEF  FRAME1
    (X+25  Y+33,5 DX+8  DY+10 NUMX5
    NUMY4 ROT+0  ROTX+0 ROTY+0  Z
    +0)
```



Defining a full circle

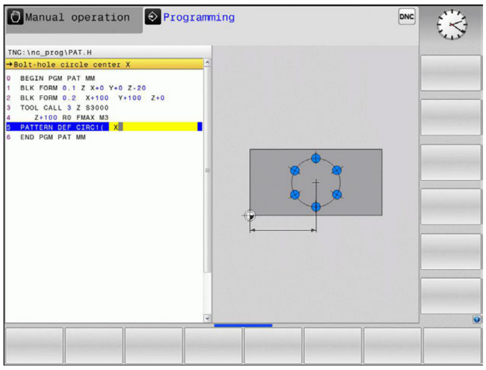
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis.
- **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis.
- **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Number of operations**: Total number of machining positions on the circle
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

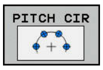
NC blocks

```
10 Z+100 R0 FMAX
11 PATTERN DEF CIRC1
   (X+25 Y+33 D80 START+45 NUM8 Z
   +0)
```



Defining a pitch circle

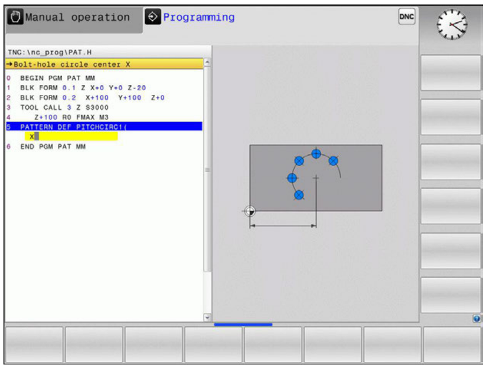
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Bolt-hole circle center X** (absolute): Coordinate of the center point of the circle in the X axis
- **Bolt-hole circle center Y** (absolute): Coordinate of the center point of the circle in the Y axis
- **Bolt-hole circle diameter**: Diameter of the bolt hole circle
- **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Stepping angle/Stopping angle**: Incremental polar angle between two machining positions. You can enter a positive or negative value. As an alternative you can enter the end angle (switch via soft key).
- **Number of operations**: Total number of machining positions on the circle
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

NC blocks

```
10 Z+100 R0 FMAX
11 PATTERN DEF PITCHCIRC1
   (X+25 Y+33 D80 START+45 STEP30
   NUM8 Z+0)
```



16.6 POLAR PATTERN (Cycle 220)

Cycle run

1 At rapid traverse, the TNC moves the tool from its current position to the starting point for the first machining operation.
  Sequence:
  - Move to the 2nd set-up clearance (spindle axis)
  - Approach the starting point in the spindle axis.
  - Move to the set-up clearance above the workpiece surface (spindle axis)
2 From this position, the TNC executes the last defined fixed cycle.
3 The tool then approaches on a straight line the starting point for the next machining operation. The tool stops at the set-up clearance (or the 2nd set-up clearance).
4 This process (1 to 3) is repeated until all machining operations have been executed.

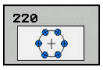
Please note while programming:

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

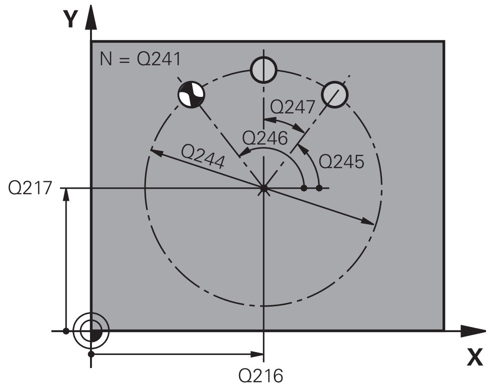
If you combine Cycle 220 with one of the fixed cycles 200 to 207 and 251, 253 and 256 or with cycle 221, the set-up clearance, workpiece surface and the 2nd set-up clearance that were defined in Cycle 220 or 221 will be effective. This applies in the program until the affected parameters are overwritten again. Example: If in a program Cycle 200 is defined with Q203=0 and then a cycle 220 is programmed with Q203=-5, Q203=-5 is used with the subsequent CYCL CALL and M99 calls. Cycles 220 and 221 overwrite the above-specified parameters of the CALL-active machining cycles (with identical input parameters in both cycles).

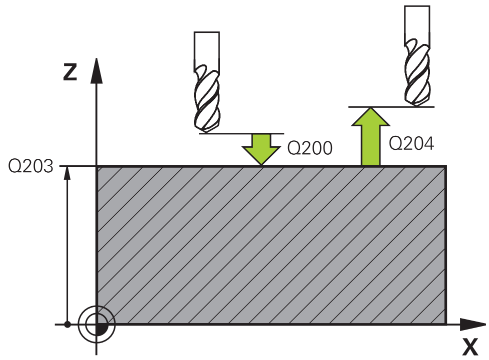
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q216 Center in 1st axis?** (absolute): Pitch circle center in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- **Q217 Center in 2nd axis?** (absolute): Pitch circle center in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- **Q244 Pitch circle diameter?**: Diameter of the pitch circle. Input range 0 to 99999.9999
- **Q245 Starting angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range -360.000 to 360.000
- **Q246 Stopping angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range -360.000 to 360.000
- **Q247 Intermediate stepping angle?** (incremental): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). Input range -360.000 to 360.000
- **Q241 Number of repetitions?**: Number of machining operations on a pitch circle. Input range 1 to 99999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the tool moves between machining operations:
  0: Move at safety clearance between machining operations
  1: Move at 2nd safety clearance between machining operations



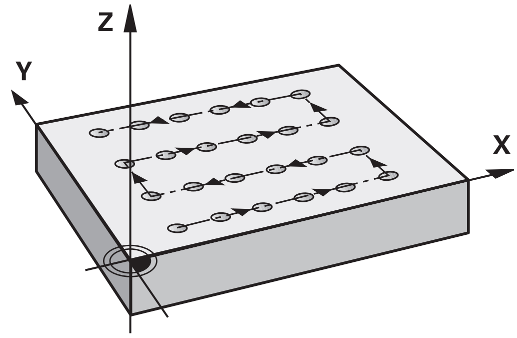


NC blocks

53 CYCL DEF 220 POLAR PATTERN	
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q244=80	;PITCH CIRCLE DIAMETR
Q245=+0	;STARTING ANGLE
Q246=+360	;STOPPING ANGLE
Q247=+0	;STEPPING ANGLE
Q241=8	;NR OF REPETITIONS
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE

16.7 LINEAR PATTERN (Cycle 221)

Cycle run



1 The TNC automatically moves the tool from its current position to the starting point for the first machining operation.
   Sequence:
   - Move to the 2nd set-up clearance (spindle axis)
   - Approach the starting point in the machining plane
   - 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 set-up clearance (or 2nd set-up clearance).
4 This process (1 to 3) is repeated until all machining operations on the first line have been executed. The tool is located above the last point on the first line.
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.
6 From this position, the tool approaches the starting point for the next machining operation in the negative 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.

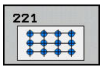
Please note while programming:

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

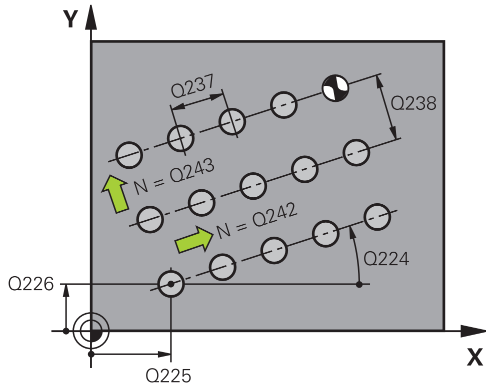
If you combine Cycle 221 with one of the fixed cycles 200 to 207 and 251, 253 and 256, the set-up clearance, workpiece surface, the 2nd set-up clearance, and the rotational position that were defined in Cycle 221 will be effective.

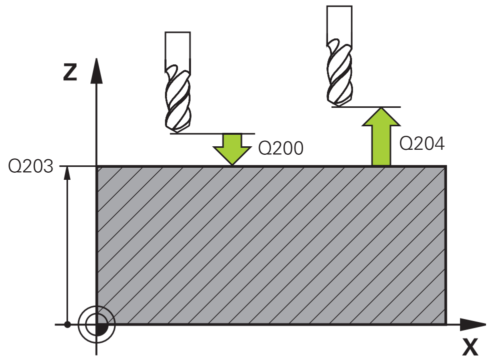
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q225 Starting point in 1st axis?** (absolute): Coordinate of the starting point in the reference axis of the working plane
- **Q226 Starting point in 2nd axis?** (absolute): Coordinate of the starting point in the minor axis of the working plane
- **Q237 Spacing in 1st axis?** (incremental): Spacing between the individual points on a line
- **Q238 Spacing in 2nd axis?** (incremental): Spacing between the individual lines
- **Q242 Number of columns?**: Number of machining operations on a line
- **Q243 Number of lines?**: Number of lines
- **Q224 Angle of rotation?** (absolute): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the tool moves between machining operations:
  0: Move at safety clearance between machining operations
  1: Move at 2nd safety clearance between machining operations





NC blocks

54 CYCL DEF 221 CARTESIAN PATTERN	
Q225=+15	;STARTNG PNT 1ST AXIS
Q226=+15	;STARTNG PNT 2ND AXIS
Q237=+10	;SPACING IN 1ST AXIS
Q238=+8	;SPACING IN 2ND AXIS
Q242=6	;NUMBER OF COLUMNS
Q243=4	;NUMBER OF LINES
Q224=+15	;ANGLE OF ROTATION
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE

16.8 Point tables

Application

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. Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Creating a point table

- Mode of operation: Press the **Programming** key

PGM MGT

- Call the file manager: Press the **PGM MGT** key.

FILE NAME?

ENT

- Enter the name and file type of the point table and confirm your entry with the **ENT** key.

MM

- 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.

> The name of the point table must begin with a letter.
>
> Use the soft keys **X OFF/ON**, **Y OFF/ON**, **Z OFF/ON** (second soft-key row) to 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.

- In the table, select the point to be hidden
- Select the **FADE** column
- Activate hiding, or
- Deactivate hiding

Selecting a point table in the program

In the **Programming** 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. If the point table is not stored in the same directory as the NC program, you must enter the complete path.

Example NC block

```
7 SEL PATTERN  "TNC:\DIRKT5\NUST35.PNT"
```

Calling a cycle in connection with point tables

With **CYCL CALL PAT** the TNC runs the point table that you last defined (even if you defined the point table in a program that was nested with **CALL PGM**).

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 **CYCLE CALL PAT**:

CYCL CALL

- To program the cycle call, press the **CYCL CALL** key
- Press the **CYCL CALL PAT** soft key to call a point table
- Enter the feed rate at which the TNC is to move from point to point (if you make no entry the TNC will move at the last programmed feed rate; **FMAX** is not valid)
- If required, enter a miscellaneous function M, then confirm with the **END** key

The TNC retracts the tool to the clearance height between the starting points. Depending on which is greater, the TNC uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.

Before **CYCL CALL PAT** you can use the function **GLOBAL DEF 125** (located in **SPEC FCT**/program defaults) with Q352=1. Then the TNC always retracts the tool between the holes to the 2nd set-up clearance that was defined in the cycle.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103.

Effect of the point table with Cycles 200 to 207

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 table with Cycles 251, 253 and 256

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.

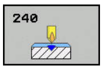
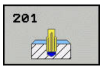
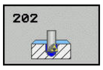
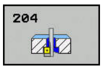
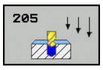
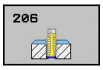
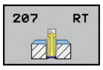
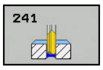
17

Cycles: Drilling cycles / thread cycles

17.1 Fundamentals

Overview

The TNC offers the following cycles for all types of drilling and threading operations:

Soft key	Cycle	Page
	240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth	525
	200 DRILLING With automatic pre-positioning, 2nd set-up clearance	527
	201 REAMING With automatic pre-positioning, 2nd set-up clearance	529
	202 BORING With automatic pre-positioning, 2nd set-up clearance	531
	203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	534
	204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	540
	205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	544
	206 TAPPING With floating tap holder, 2nd set-up clearance, dwell time at depth	561
	207 RIGID TAPPING With thread depth and thread pitch	563
	241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition	551

17.2 CENTERING (Cycle 240)

Cycle run

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 programmed centering diameter or centering depth.
3 If defined, the tool remains at the centering depth.
4 Finally, the tool path is retraced to setup clearance or—if programmed—to the 2nd setup clearance at rapid traverse **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with the radius compensation **R0**

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.

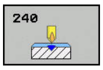
NOTICE

Danger of collision!

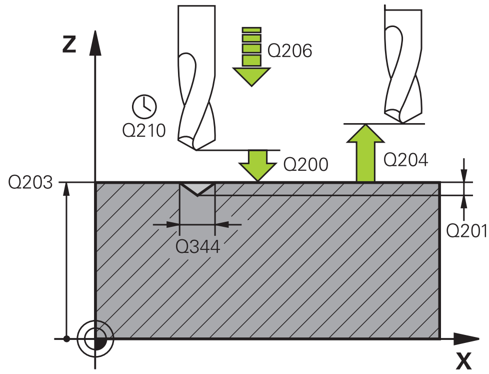
If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

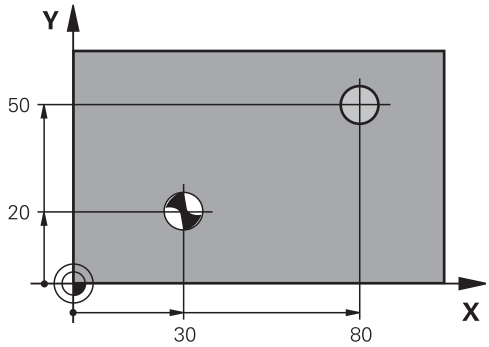
- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- **Q343 Select diameter/depth (1/0)**: Select whether centering is based on the entered diameter or entered depth. If the TNC is to center 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.
  0: Centering based on the entered depth
  1: Centering based on the entered diameter
- **Q201 Depth?** (incremental): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined. Input range -99999.9999 to 99999.9999
- **Q344 Diameter of counterbore** (algebraic sign): Centering diameter. Only effective if Q343=1 is defined. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while centering. Input range 0 to 99999.999, alternatively **FAUTO**, **fu**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





NC blocks

```
11 CYCL DEF 240 CENTERING
    Q200=2      ;SET-UP CLEARANCE
    Q343=1      ;SELECT DIA./DEPTH
    Q201=+0     ;DEPTH
    Q344=-9     ;DIAMETER
    Q206=250    ;FEED RATE FOR PLNGNG
    Q211=0.1    ;DWELL TIME AT DEPTH
    Q203=+20    ;SURFACE COORDINATE
    Q204=100    ;2ND SET-UP CLEARANCE
12 X+30  R0 FMAX
13 Y+20  R0 FMAX  M3 M99
14 X+80  R0 FMAX
15 Y+50  R0 FMAX  M99
```

17.3 DRILLING (Cycle 200)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to set-up clearance above the workpiece surface.
2. The tool drills to the first plunging depth at the programmed feed rate **F**.
3. The TNC returns the tool at **FMAX** to the set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to the set-up clearance above the first plunging depth.
4. The tool then drills deeper by the plunging depth at the programmed feed rate F.
5. The TNC repeats this process (2 to 4) until the programmed depth is reached (the dwell time from Q211 is effective with every infeed)
6. Finally, the tool path is retraced to setup clearance from the hole bottom or—if programmed—to the 2nd setup clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**

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

NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

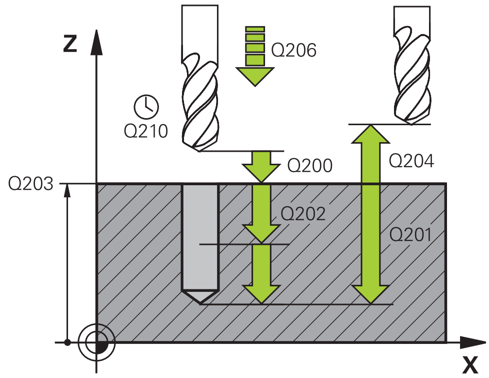
- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

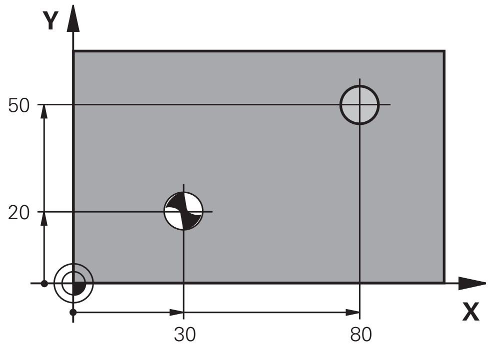
Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- **Q202 Plunging depth?** (incremental): Infeed per cut Input range 0 to 99999.9999

  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
- **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip breaking. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the tool table TOOL.T.
  0 = Depth referenced to the tool tip
  1 = Depth referenced to the cylindrical part of the tool





NC blocks

11 CYCL DEF 200 DRILLING
Q200=2 ;SET-UP CLEARANCE
Q201=-15 ;DEPTH
Q206=250 ;FEED RATE FOR PLNGNG
Q202=5 ;PLUNGING DEPTH
Q211=0 ;DWELL TIME AT TOP
Q203=+20 ;SURFACE COORDINATE
Q204=100 ;2ND SET-UP CLEARANCE
Q211=0.1 ;DWELL TIME AT DEPTH
Q395=0 ;DEPTH REFERENCE
12 X+30 FMAX
13 Y+20 FMAX M3 M99
14 X+80 FMAX
15 Y+50 FMAX M99

17.4 REAMING (Cycle 201)

Cycle run

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

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

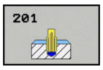
NOTICE

Danger of collision!

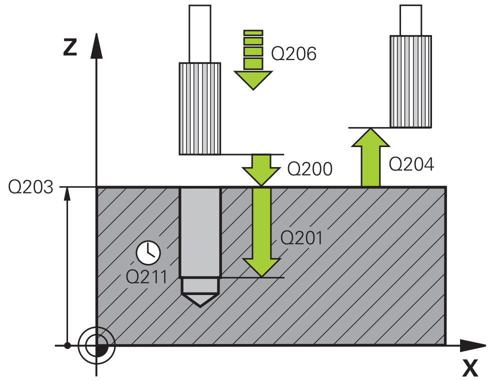
If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

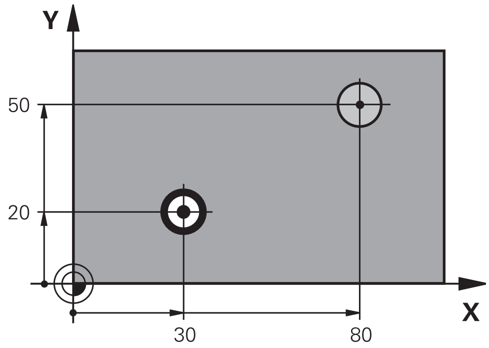
- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while reaming. Input range 0 to 99999.999, alternatively **FAUTO**, **fu**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q208 Feed rate for retraction?**: Traverse speed of tool when moving out of the hole in mm/min. If you enter Q208 = 0, the feed rate for reaming applies. Input range 0 to 99999.999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range 0 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





NC blocks

```
11 CYCL DEF 201 REAMING
   Q200=2      ;SET-UP CLEARANCE
   Q201=-15    ;DEPTH
   Q206=100    ;FEED RATE FOR PLNGNG
   Q211=0.5    ;DWELL TIME AT DEPTH
   Q208=250    ;RETRACTION FEED RATE
   Q203=+20    ;SURFACE COORDINATE
   Q204=100    ;2ND SET-UP CLEARANCE
12 X+30  FMAX
13 Y+20  FMAX M3 M99
14  X+80 FMAX
15 Y+50  FMAX M9
```

17.5 BORING (Cycle 202)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to set-up clearance above the workpiece surface.
2 The tool drills to the programmed depth at the feed rate for plunging.
3 If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4 The TNC then orients the spindle to the 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 tool then retracts to set-up clearance at the retraction rate, and from there—if programmed—to the 2nd set-up clearance at **FMAX**. If Q214=0 the tool point remains on the wall of the hole.
7 The TNC finally positions the tool back at the center of the hole.

Please note while programming:

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

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

After machining, the TNC positions the tool back at the starting point of the machining plane. This way, you can continue positioning incrementally.

If the functions M7 or M8 were active before calling the cycle, the TNC will reconstruct this previous state at the end of the cycle.

NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

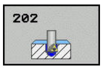
NOTICE

Danger of collision!

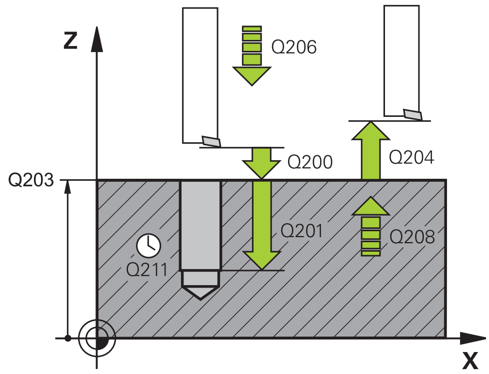
There is a danger of collision if you select the disengaging direction incorrectly. Any existing mirroring in the work plane is not taken into account for the disengaging direction. However, active transformations are considered with disengaging.

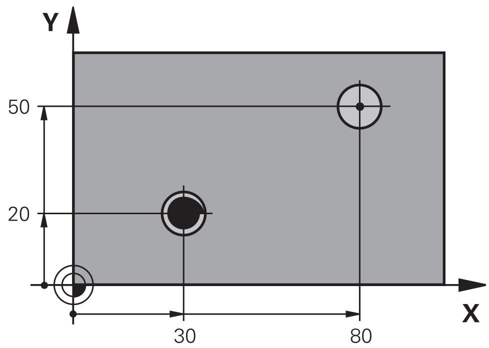
- Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the **Positioning with Manual Data Input** mode of operation). No transformations should be active here.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while boring. Input range 0 to 99999.999, alternatively **FAUTO**, **fu**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q208 Feed rate for retraction?**: Traverse speed of tool when moving out of the hole in mm/min. If you enter Q208 = 0, the feed rate for plunging applies. Input range 0 to 99999.999, alternatively **Fmax**, **FAUTO**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q214 Disengaging directn (0/1/2/3/4)?**: Determine the direction in which the TNC disengages the tool at the hole bottom (after the spindle orientation)
  0: Do not disengage the tool
  1: Disengage the tool in a minus direction of the reference axis
  2: Disengage the tool in a minus direction of the minor axis
  3: Disengage the tool in a plus direction of the reference axis
  4: Disengage the tool in a plus direction of the minor axis
- **Q336 Angle for spindle orientation?** (absolute): Angle to which the TNC positions the tool before retracting it. Input range -360.000 to 360.000






```
10 Z+100 R0 FMAX
11 CYCL DEF 202 BORING
   Q200=2      ;SET-UP CLEARANCE
   Q201=-15    ;DEPTH
   Q206=100    ;FEED RATE FOR PLNGNG
   Q211=0.5    ;DWELL TIME AT DEPTH
   Q208=250    ;RETRACTION FEED RATE
   Q203=+20    ;SURFACE COORDINATE
   Q204=100    ;2ND SET-UP CLEARANCE
   Q214=1      ;DISENGAGING DIRECTN
   Q336=0      ;ANGLE OF SPINDLE
12 X+30 FMAX
13 Y+20 FMAX M3 M99
14 X+80 FMAX
14 Y+50 FMAX M99
```

17.6 UNIVERSAL DRILLING (Cycle 203)

Cycle run

Behavior without chip breaking and without decrement:

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the programmed **SET-UP CLEARANCEQ200** above the workpiece surface
2 The tool drills at the programmed **FEED RATE FOR PLNGNGQ206** to the first **PLUNGING DEPTHQ202**
3 Then the TNC removes the tool from the hole to **SET-UP CLEARANCEQ200**
4 The TNC now again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTHQ202 FEED RATE FOR PLNGNGQ206**
5 When machining without chip breakage the TNC removes the tool from the hole after each infeed with **RETRACTION FEED RATEQ208** to **SET-UP CLEARANCEQ200** and remains there for the **DWELL TIME AT TOPQ210**.
6 This procedure is repeated until **depth Q201** is achieved.
7 When **depth Q201** is achieved, the TNC removes the tool with **Fmax** from the hole to the **2nd set-up clearance Q204**

Behavior with chip breaking and without decrement:

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool drills at the programmed **feed rate for plunging Q206** to the first **plunging depth Q202**
3. The TNC then disengages the tool by the value of **Retraction rate for chip breaking Q256**
4. An infeed is then again machined by the value of **plunging depth Q202** at **feed rate for plunging Q206**
5. The TNC repeatedly infeeds until achieving the **number of chip breaks Q213** or until the hole has the desired **depth Q201**.
   If the defined number of chip breaks is achieved but the hole does not yet have the desired **depth Q201**, the TNC retracts the tool from the hole at **feed rate for retraction Q208** to **set-up clearance Q200**
6. If programmed, the TNC now waits in accordance with the **dwell time at top Q210**
7. Then the TNC plunges at rapid traverse into the hole to the value **retraction rate for chip breaking Q256** above the last infeed depth
8. Procedure 2-7 is repeated until **depth Q201** is achieved.
9. When **Depth Q201** is achieved, the TNC removes the tool with **Fmax** from the hole to the **2nd set-up clearance Q204**

Behavior with chip breaking and with decrement

1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface
2 The tool drills at the programmed **feed rate for plunging Q206** to the first **plunging depth Q202**
3 The TNC then disengages the tool by the value of **Retraction rate for chip breaking Q256**
4 An infeed is then again machined by the value of **plunging depth Q202** minus **decrement Q212** at **feed rate for plunging Q206**. The continuously declining difference from the updated **plunging depth Q202** minus **decrement Q212** must never be less than the **minimum plunging depth Q205** (example: Q202=5, Q212=1, Q213=4, Q205= 3: The first plunging depth is 5 mm, the second plunging depth is 5 - 1 = 4 mm, the third plunging depth is 4 - 1 = 3 mm and the fourth plunging depth is also 3 mm)
5 The TNC repeatedly infeeds until achieving the **number of chip breaks Q213** or until the hole has the desired **depth Q201**. If the defined number of chip breaks is achieved but the hole does not yet have the desired **depth Q201**, the TNC retracts the tool from the hole at **feed rate for retraction Q208** to **set-up clearance Q200**
6 If programmed, the TNC now waits in accordance with the **dwell time at top Q210**
7 Then the TNC plunges at rapid traverse into the hole to the value **retraction rate for chip breaking Q256** above the last infeed depth
8 Procedure 2-7 is repeated until **depth Q201** is achieved.
9 If programmed, the TNC now waits in accordance with the **dwell time at depth Q211**
10 When **Depth Q201** is achieved and **dwell time at depth Q211** has expired, the TNC removes the tool with **Fmax** from the hole to the **2nd set-up clearance Q204**

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

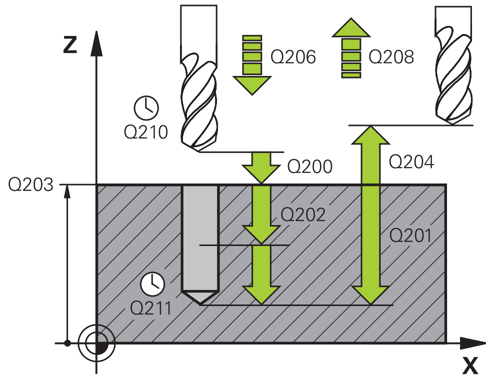
- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- **Q202 Plunging depth?** (incremental): Infeed per cut Input range 0 to 99999.9999

  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
- **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip breaking. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the TNC decreases **Q202 MAX. PLUNGING DEPTH** after each infeed. Input range 0 to 99999.9999
- **Q213 Nr of breaks before retracting?**: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip breaking. For chip breaking, the TNC retracts the tool each time by the value in Q256. Input range 0 to 99999
- **Q205 Minimum plunging depth?** (incremental): If you have programmed **Q212 DECREMENT** the TNC limits the infeed to **Q205**. Input range 0 to 99999.9999



NC blocks

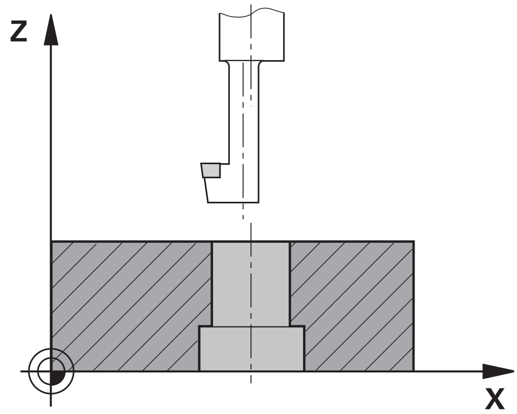
11 CYCL DEF 203 UNIVERSAL DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q211=0	;DWELL TIME AT TOP
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q212=0.2	;DECREMENT
Q213=3	;NR OF BREAKS
Q205=3	;MIN. PLUNGING DEPTH
Q211=0.25	;DWELL TIME AT DEPTH
Q208=500	;RETRACTION FEED RATE
Q256=0.2	;DIST FOR CHIP BRKNG
Q395=0	;DEPTH REFERENCE

- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q208 Feed rate for retraction?**: Traverse speed of tool when moving out of the hole in mm/min. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.999, alternatively **Fmax**, **FAUTO**
- **Q256 Retract dist. for chip breaking?** (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the tool table TOOL.T.
  0 = Depth referenced to the tool tip
  1 = Depth referenced to the cylindrical part of the tool

17.7 BACK BORING (Cycle 204)

Cycle run

This cycle allows holes to be bored from the underside of the workpiece.



1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to set-up clearance above the workpiece surface.
2 The TNC then orients the spindle to the 0° position with an oriented spindle stop and displaces the tool by the off-center distance.
3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached set-up clearance on the underside of the workpiece.
4 The TNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.
5 If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. The TNC carries out another oriented spindle stop and the tool is once again displaced by the off-center distance.
6 The tool then retracts to set-up clearance at the feed rate for pre-positioning, and from there—if programmed—to the 2nd set-up clearance at **FMAX**.
7 The TNC finally positions the tool back at the center of the hole.

Please note while programming:

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

This cycle is effective only for machines with servo-controlled spindle.

Special boring bars for upward cutting are required for this cycle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

After machining, the TNC positions the tool back at the starting point of the machining plane. This way, you can continue positioning incrementally.

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.

Enter the tool length so that the underside of the boring bar is measured and not the tool tip.

When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.

If the functions M7 or M8 were active before calling the cycle, the TNC will reconstruct this previous state at the end of the cycle.

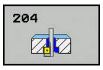
NOTICE

Danger of collision!

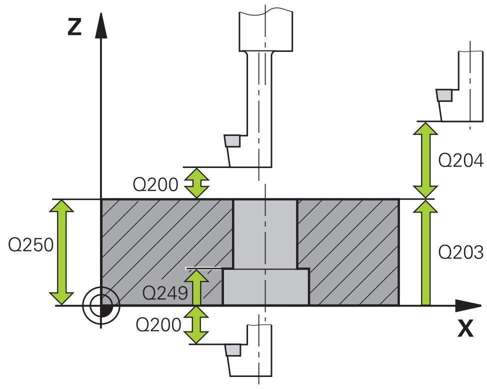
There is a danger of collision if you select the disengaging direction incorrectly. Any existing mirroring in the work plane is not taken into account for the disengaging direction. However, active transformations are considered with disengaging.

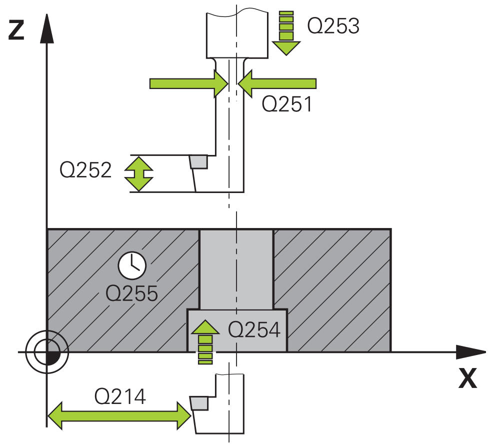
- Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the **Positioning with Manual Data Input** mode of operation). No transformations should be active here.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q249 Depth of counterbore?** (incremental): Distance between underside of workpiece and the top of hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range -99999.9999 to 99999.9999
- **Q250 Material thickness?** (incremental): Thickness of the workpiece. Input range 0.0001 to 99999.9999
- **Q251 Tool edge off-center distance?** (incremental): Off-center distance for the boring bar; value from the tool data sheet. Input range 0.0001 to 99999.9999
- **Q252 Tool edge height?** (incremental): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet. Input range 0.0001 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **fmax**, **FAUTO**
- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **fu**
- **Q255 Dwell time in secs.?**: Dwell time at counterbore floor. Input range 0 to 3600.000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





NC blocks

11 CYCL DEF 204 BACK BORING	
Q200=2	**;SET-UP CLEARANCE**
Q249=+5	**;DEPTH OF COUNTERBORE**
Q250=20	**;MATERIAL THICKNESS**
Q251=3.5	**;OFF-CENTER DISTANCE**
Q252=15	**;TOOL EDGE HEIGHT**
Q253=750	**;F PRE-POSITIONING**

- **Q214 Disengaging directn (0/1/2/3/4)?**: Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation); programming 0 is not allowed
  1: Retract the tool in negative direction of the principle axis
  2: Retract the tool in negative direction of the minor axis
  3: Retract the tool in positive direction of the principle axis
  4: Retract the tool in positive direction of the minor axis
- **Q336 Angle for spindle orientation?** (absolute): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole. Input range -360.0000 to 360.0000

Q254=200	;F COUNTERBORING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE

17.8 UNIVERSAL PECKING (Cycle 205)

Cycle run

1 The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2 If you enter a deepened starting point, the TNC move at the defined positioning feed rate to the set-up clearance above the deepened starting point.
3 The tool drills to the first plunging depth at the entered feed rate **F**.
4 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered starting position above the first plunging depth.
5 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
6 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
7 The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

If you enter different advance stop distances for **Q258** and **Q259**, the TNC will change the advance stop distances between the first and last plunging depths at the same rate.

If you use **Q379** to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. The TNC does not change retracting movements; the are referenced to the coordinate of the workpiece surface.

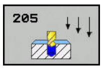
NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

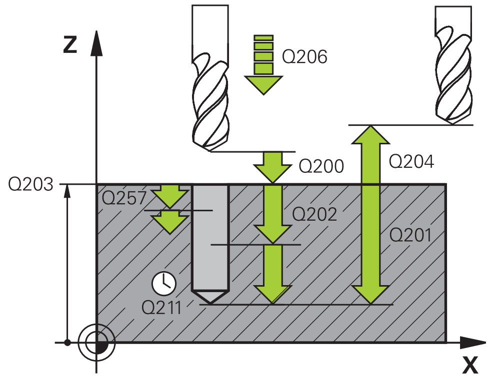
Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- **Q202 Plunging depth?** (incremental): Infeed per cut Input range 0 to 99999.9999

  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
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the TNC decreases the plunging depth Q202. Input range 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have programmed **Q212 DECREMENT** the TNC limits the infeed to **Q205**. Input range 0 to 99999.9999
- **Q258 Upper advanced stop distance?** (incremental): Setup clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole. Input range 0 to 99999.9999
- **Q259 Lower advanced stop distance?** (incremental): Lower advanced stop distance Q259 (incremental): 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. Input range 0 to 99999.9999
- **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the TNC breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Q256 Retract dist. for chip breaking?** (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.000 to 99999.999



NC blocks

11 CYCL DEF 205 UNIVERSAL PECKING	
Q200=2	**;SET-UP CLEARANCE**
Q201=-80	**;DEPTH**
Q206=150	**;FEED RATE FOR PLNGNG**
Q202=15	**;PLUNGING DEPTH**
Q203=+100	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**
Q212=0.5	**;DECREMENT**
Q205=3	**;MIN. PLUNGING DEPTH**
Q258=0.5	**;UPPER ADV STOP DIST**
Q259=1	**;LOWER ADV STOP DIST**
Q257=5	**;DEPTH FOR CHIP BRKNG**
Q256=0.2	**;DIST FOR CHIP BRKNG**
Q211=0.25	**;DWELL TIME AT DEPTH**
Q379=7.5	**;STARTING POINT**
Q253=750	**;F PRE-POSITIONING**
Q208=9999	**;RETRACTION FEED RATE**
Q395=0	**;DEPTH REFERENCE**

- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q379 Deepened starting point?** (incremental with respect to **Q203 SURFACE COORDINATE**, takes Q200 into account): Starting position of actual drilling. The TNC moves at **Q253 F PRE-POSITIONING** to the value **Q200 SET-UP CLEARANCE** above the deepened starting point. Input range 0 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when returning to **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Entry in mm/min. Input range 0 to 99999.9999 alternatively **fmax**, **FAUTO**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. Input range 0 to 99999.9999, alternatively **fmax,FAUTO**
- **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the TNC is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the tool table TOOL.T.
  0 = Depth referenced to the tool tip
  1 = Depth referenced to the cylindrical part of the tool

Positioning behavior during program run with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. With overlong drills, this can lead to tool breakage if the necessary guidance of the tool is lacking.

It is therefore advisable to use the parameter **STARTING POINT Q379**. This parameter can be used to influence the position at which the TNC turns on the spindle.

Starting point for drilling

The **STARTING POINT Q379** parameter considers the **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The TNC switches the spindle on at the **SET-UP CLEARANCE Q200** via the **SURFACE COORDINATE Q203** .

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point Q379. This value is calculated as follows: **0.2 x Q379** If the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2
- The starting point of drilling is calculated as follows: 0.2 x Q379=0.2*2=0.4; the starting point is 0.4 mm/inch over the recessed starting point. So if the recessed starting point is at -2, the TNC starts the drilling process at -1.6 mm.

  The following table lists various examples of how the starting point is calculated:

Drilling start at the recessed starting point

Q200	Q379	Q203	Position, to which you pre-position with FMAX	Factor of 0.2 * Q379	Starting point for drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (Q200=2, 5>2, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (Q200=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (Q200=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

The point at which the TNC performs the removal process also plays a decisive role for the work with overlong tools. The retraction position during the removal process does not have to be at the position of the drilling start. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed at the **SET-UP CLEARANCE Q200** over the **SURFACE COORDINATE Q203** .

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point Q379. This value is calculated as follows: **0.8 x Q379** If the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2
- The position for chip removal is calculated as follows: 0.8 x Q379 = 0.8 * 2 = 1.6; the position chip removal is 1.6 mm/inch above the recessed start point. So if the recessed starting point is at -2, the TNC starts the chip removal process at -0.4 mm.

The following table shows examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with recessed starting point

Q200	Q379	Q203	Position, to which you pre-position with FMAX	Factor of 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (Q200=2, 8>2, so the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (Q200=2, 20>2, so the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (Q200=2, 80>2, so the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (Q200=5, 8>5, so the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (Q200=5, 20>5, so the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (Q200=5, 80>5, so the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (Q200=20, 80>20, so the value 20 is used.)	-80

17.9 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)

Cycle run

1. The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the programmed **Safety clearance Q200** above the **SURFACE COORDINATE Q203**
2. Depending on the "Positioning behavior during program run with Q379", page 548, the TNC switches the spindle speed either to the **Safety clearance Q200** or to a specific value above the coordinate surface. see page 548
3. The TNC executes the approach motion with the direction of rotation defined in the cycle, with clockwise, counterclockwise or stationary spindle.
4. The tool drills to the hole depth at the feed rate **F**, or to the plunging depth if a smaller infeed value has been entered. The plunging depth is decreased after each infeed by the decrement. If you have entered a dwell depth, the TNC reduces the feed rate by the feed rate factor after the dwell depth has been reached.
5. If programmed, the tool remains at the hole bottom for chip breaking.
6. The TNC repeats this process (4 to 5) until the hole depth is reached.
7. After the TNC has reached the hole depth, it switches off the coolant and resets the drilling speed to the value defined in Q427 **ROT.SPEED INFEED/OUT**.
8. The TNC positions the tool at the retraction feed rate to the retraction position. Refer to the following document for the value of the retraction position in your case: see page 548
9. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

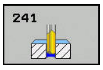
NOTICE

Danger of collision!

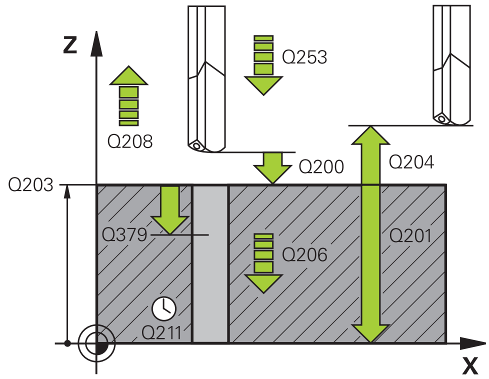
If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance of tool tip to **Q203 SURFACE COORDINATE**. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance of **Q203 SURFACE COORDINATE** to bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Distance to workpiece datum. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q379 Deepened starting point?** (incremental with respect to **Q203 SURFACE COORDINATE**, takes Q200 into account): Starting position of actual drilling. The TNC moves at **Q253 F PRE-POSITIONING** to the value **Q200 SET-UP CLEARANCE** above the deepened starting point. Input range 0 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when returning to **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Entry in mm/min. Input range 0 to 99999.9999 alternatively **fmax**, **FAUTO**
- **Q208 Feed rate for retraction?**: Traverse speed of tool when moving out of the hole in mm/min. If you enter **Q208**=0, the TNC retracts the tool at **Q206 FEED RATE FOR PLNGNG**. Input range 0 to 99999.999, alternatively **Fmax**, **FAUTO**
- **Q426 Rot. dir. of entry/exit (3/4/5)?**: Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input:
  3: Turn the spindle with M3
  4: Turn the spindle with M4
  5: Move with stationary spindle
- **Q427 Spindle speed of entry/exit?**: Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input range 0 to 99999



NC blocks

```
11 CYCL DEF 241 SINGLE-LIP
   D.H.DRLNG
   Q200=2      ;SET-UP CLEARANCE
   Q201=-80    ;DEPTH
   Q206=150    ;FEED RATE FOR PLNGNG
   Q211=0.25   ;DWELL TIME AT DEPTH
   Q203=+100   ;SURFACE COORDINATE
   Q204=50     ;2ND SET-UP CLEARANCE
   Q379=7.5    ;STARTING POINT
   Q253=750    ;F PRE-POSITIONING
   Q208=1000   ;RETRACTION FEED RATE
   Q426=3      ;DIR. OF SPINDLE ROT.
   Q427=25     ;ROT.SPEED INFEED/OUT
   Q428=500    ;ROT. SPEED DRILLING
   Q429=8      ;COOLANT ON
   Q430=9      ;COOLANT OFF
   Q435=0      ;DWELL DEPTH
   Q401=100    ;FEED RATE FACTOR
   Q202=9999   ;MAX. PLUNGING DEPTH
   Q212=0      ;DECREMENT
   Q205=0      ;MIN. PLUNGING DEPTH
```

- **Q428 Spindle speed for drilling?**: Desired speed for drilling. Input range 0 to 99999
- **Q429 M function for coolant on?**: Miscellaneous function M for switching on the coolant. The TNC switches the coolant on if the tool is in the hole at **Q379 STARTING POINT**. Input range 0 to 999
- **Q430 M function for coolant off?**: Miscellaneous function M for switching off the coolant. The TNC switches the coolant off if the tool is at **Q201 DEPTH**. Input range 0 to 999
- **Q435 Dwell depth?** (incremental): Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (standard setting). Application: During machining of through-holes some tools require a short dwell time before exiting the bottom of the hole in order to transport the chips to the top. Define a value smaller than **Q201 DEPTH**, input range 0 to 99999.9999.
- **Q401 Feed rate factor in %?**: Factor by which the TNC reduces the feed rate after the **Q435 DWELL DEPTH** has been reached. Input range 0 to 100
- **Q202 Maximum plunging depth?** (incremental): Infeed per cut **Q201 DEPTH** does not have to be a multiple of **Q202**. Input range 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the TNC decreases **Q202 MAX. PLUNGING DEPTH** after each infeed. Input range 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have programmed **Q212 DECREMENT** the TNC limits the infeed to **Q205**. Input range 0 to 99999.9999

Positioning behavior during program run with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. With overlong drills, this can lead to tool breakage if the necessary guidance of the tool is lacking.

It is therefore advisable to use the parameter **STARTING POINT Q379**. This parameter can be used to influence the position at which the TNC turns on the spindle.

Starting point for drilling

The **STARTING POINT Q379** parameter considers the **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The TNC switches the spindle on at the **SET-UP CLEARANCE Q200** via the **SURFACE COORDINATE Q203** .

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point Q379. This value is calculated as follows: **0.2 x Q379** If the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2
- The starting point of drilling is calculated as follows: 0.2 x Q379=0.2*2=0.4; the starting point is 0.4 mm/inch over the recessed starting point. So if the recessed starting point is at -2, the TNC starts the drilling process at -1.6 mm.

  The following table lists various examples of how the starting point is calculated:

Drilling start at the recessed starting point

Q200	Q379	Q203	Position, to which you pre-position with FMAX	Factor of 0.2 * Q379	Starting point for drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (Q200=2, 5>2, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (Q200=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (Q200=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

The point at which the TNC performs the removal process also plays a decisive role for the work with overlong tools. The retraction position during the removal process does not have to be at the position of the drilling start. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed at the **SET-UP CLEARANCE Q200** over the **SURFACE COORDINATE Q203** .

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point Q379. This value is calculated as follows: **0.8 x Q379** If the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2
- The position for chip removal is calculated as follows: 0.8 x Q379 = 0.8 * 2 = 1.6; the position chip removal is 1.6 mm/ inch above the recessed start point. So if the recessed starting point is at -2, the TNC starts the chip removal process at -0.4 mm.

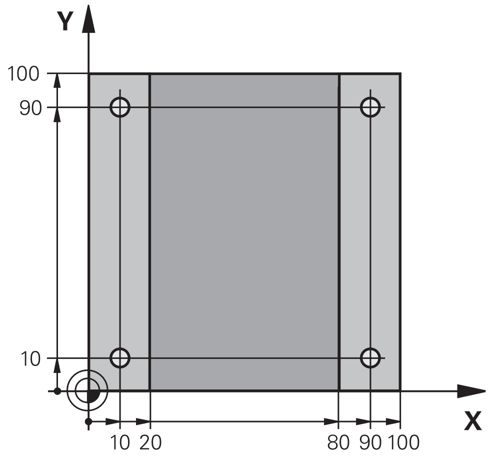
The following table shows examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with recessed starting point

Q200	Q379	Q203	Position, to which you pre-position with FMAX	Factor of 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (Q200=2, 8>2, so the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (Q200=2, 20>2, so the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (Q200=2, 80>2, so the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (Q200=5, 8>5, so the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (Q200=5, 20>5, so the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (Q200=5, 80>5, so the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (Q200=20, 80>20, so the value 20 is used.)	-80

17.10 Programming Examples

Example: Drilling cycles



0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4500	Tool call (tool radius 3)
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition
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	
Q395=0 ;DEPTH REFERENCE	
6 X+10 R0 FMAX M3	Approach hole 1, spindle ON
7 Y+10 R0 FMAX M99	Approach hole 1, call cycle
8 X+90 R0 FMAX M99	Approach hole 2, call cycle
9 Y+90 R0 FMAX M99	Approach hole 3, call cycle
10 X+10 R0 FMAX M99	Approach hole 4, call cycle
11 Z+250 R0 FMAX M2	Retract the tool, end program
12 END PGM C200 MM	

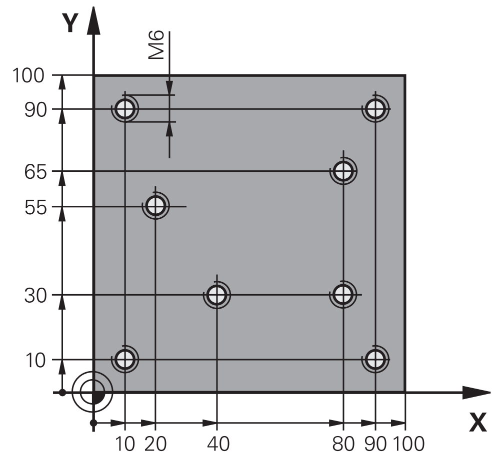
Example: Using drilling cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the pattern definition PATTERN DEF POS and are called by the TNC with CYCL CALL PAT.

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering (tool radius 4)
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL CALL 1 Z S5000	Call the centering tool (tool radius 4)
4 Z+50 R0 FMAX	Move tool to clearance height
5 PATTERN DEF	Define all drilling positions in the point pattern
POS1(X+10 Y+10 Z+0)	
POS2(X+40 Y+30 Z+0)	
POS3(X+20 Y+55 Z+0)	
POS4(X+10 Y+90 Z+0)	
POS5(X+90 Y+90 Z+0)	
POS6(X+80 Y+65 Z+0)	
POS7(X+80 Y+30 Z+0)	
POS8(X+90 Y+10 Z+0)	
6 CYCL DEF 240 CENTERING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q343=0 ;SELECT DIA./DEPTH	
Q201=-2 ;DEPTH	
Q344=-10 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
POSITION 7 GLOBAL DEF 125	With this function the TNC positions to the 2nd set-up clearance with CYCL CALL PAT between the points. This function is in effect until M30.
Q345=+1 ;SELECT POS. HEIGHT	
7 CYCL CALL PAT F5000 M13	Call the cycle in connection with the hole pattern

8 Z+100 R0 FMAX	Retract the tool, change the tool
9 TOOL CALL 2 Z S5000	Call the drilling tool (radius 2.4)
10 Z+50 R0 F5000	Move tool to clearance height
11 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
12 CYCL CALL PAT F500 M13	Call the cycle in connection with the hole pattern
13 Z+100 R0 FMAX	Retract the tool
14 TOOL CALL Z S200	Call the tapping tool (radius 3)
15 Z+50 R0 FMAX	Move tool to clearance height
16 CYCL DEF 206 TAPPING NEW	Cycle definition for tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
17 CYCLE CALL PAT F5000 M13	Call the cycle in connection with the hole pattern
18 Z+100 R0 FMAX M2	Retract the tool, end program
19 END PGM 1 MM	

17.11 TAPPING with a floating tap holder (Cycle 206)

Cycle run

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
4. At the set-up clearance, the direction of spindle rotation reverses once again.

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

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

It is possible to use the feed rate potentiometer during tapping. The machine tool builder sets the configuration (with parameter **CfgThreadSpindle>sourceOverride**) for this purpose. The TNC then modifies the speed accordingly.

The spindle speed potentiometer is inactive.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC displays an error message if the values do not match. In Cycle 206 the TNC uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

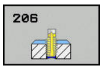
NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

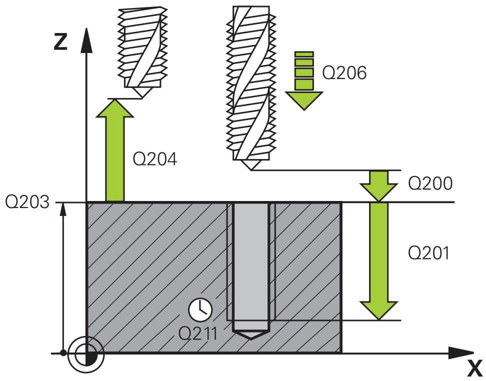
Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

Guide value: 4x pitch.

- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool during tapping. Input range 0 to 99999.999 alternatively **FAUTO**
- **Q211 Dwell time at the depth?**: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



NC blocks

25 CYCL DEF 206 TAPPING NEW	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q206=150	;FEED RATE FOR PLNGNG
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.

17.12 RIGID TAPPING without a floating tap holder (Cycle 207)

Cycle run

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

1. The TNC positions the tool in the tool axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
2. The tool drills to the total hole depth in one movement.
3. It then reverses the direction of spindle rotation again and the tool is retracted to the setup clearance. If you have entered a 2nd set-up clearance the TNC will move the tool with **FMAX** towards it.
4. The TNC stops the spindle turning at set-up clearance.

Please note while programming:

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

This cycle is effective only for machines with servo-controlled spindle.

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

It is possible to use the feed rate potentiometer during tapping. The machine tool builder sets the configuration (with parameter **CfgThreadSpindle>sourceOverride**) for this purpose. The TNC then modifies the speed accordingly.

The spindle speed potentiometer is inactive.

If you program M3 (or M4) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the TOOL CALL block).

If you do not program M3 (or M4) before this cycle, the spindle stands still after the end of the cycle. Then you must restart the spindle with M3 (or M4) before the next operation.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the TNC compares the thread pitch from the tool table with the thread pitch defined in the cycle. The TNC displays an error message if the values do not match.

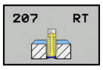
NOTICE

Danger of collision!

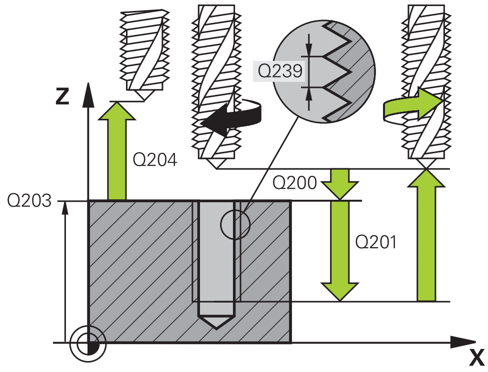
If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and root of thread. Input range -99999.9999 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  –= left-hand thread
  Input range -99.9999 to 99.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



NC blocks

26 CYCL DEF 207 RIGID TAPPING NEW	
Q200=2	**;SET-UP CLEARANCE**
Q201=-20	**;DEPTH OF THREAD**
Q239=+1	**;THREAD PITCH**
Q203=+25	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**

Retracting after a program interruption

Retracting in the Manual Operation mode

You can interrupt the thread cutting process by pressing the NC Stop key. A soft key for retracting the tool from the thread is displayed in the soft-key row below the screen. When you press this soft key and the NC Start key, the tool retracts from the hole and returns to the starting point of machining. The spindle is stopped automatically and the TNC displays a message.

Retracting in the Program Run, Single Block or Full Sequence mode

You can interrupt the thread cutting process by pressing the NC Stop key. The TNC shows the soft key **MANUAL TRAVERSE**. After pressing **MANUAL TRAVERSE** you can retract the tool in the active spindle axis. To resume machining after the interruption, press the **RESTORE POSITION** soft key and NC Start. The TNC moves the tool back to the position it had assumed before the NC Stop key was pressed.

NOTICE

Danger of collision!

A danger of collision exists if you move the tool during retraction in a negative direction instead of e.g. in a positive direction.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

17.13 Programming Examples

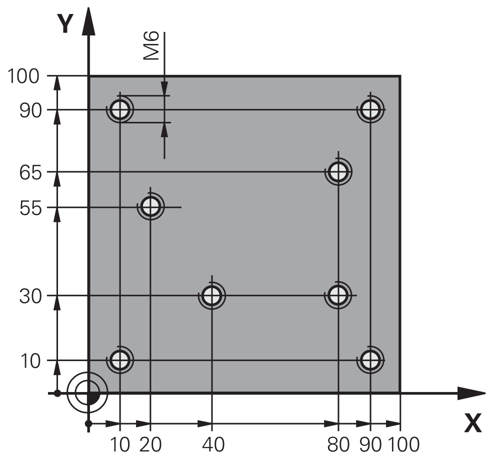
Example: Thread milling

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the TNC with **CYCL CALL PAT**.

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Call tool: centering drill
4 Z+10 R0 F5000	Move tool to clearance height (enter a value for F): the TNC positions to the clearance height after every cycle
5 SEL PATTERN “TAB1“	Definition of point table
6 CYCL DEF 240 CENTERING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q343=1 ;SELECT DIA./DEPTH	
Q201=-3.5 ;DEPTH	
Q344=-7 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q11=0 ;DWELL TIME AT DEPTH	
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
10 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT, feed rate between the points: 5000 mm/min
11 Z+100 R0 FMAX M6	Retract the tool, change the tool
12 TOOL CALL 2 Z S5000	Call tool: drill
13 Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14 CYCL DEF 200 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	
Q395=0	;DEPTH REFERENCE	
15 CYCL CALL PAT F5000 M3		Cycle call in connection with point table TAB1.PNT
16 Z+100 R0 FMAX M6		Retract the tool, change the tool
17 TOOL CALL 3 Z S200		Call tool: tap
18 Z+50 R0 FMAX		Move tool to clearance height
19 CYCL DEF 206 TAPPING		Cycle definition for tapping
Q200=2	;SET-UP CLEARANCE	
Q201=-25	;DEPTH OF THREAD	
Q206=150	;FEED RATE FOR PLNGNG	
Q211=0	;DWELL TIME AT DEPTH	
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
20 CYCL CALL PAT F5000 M3		Cycle call in connection with point table TAB1.PNT
21 Z+100 R0 FMAX M2		Retract the tool, end program
22 END PGM 1 MM		

Point table TAB1.PNT

TAB1. PNTMM
NRXYZ
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]

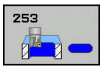
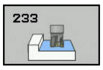
18

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

18.1 Fundamentals

Overview

The TNC offers the following cycles for machining pockets, studs and slots:

Soft key	Cycle	Page
	251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation	571
	253 SLOT MILLING Roughing/finishing cycle with selection of machining operation	575
	256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	579
	233 FACE MILLING Machining the face with up to 3 limits	583

18.2 RECTANGULAR POCKET (Cycle 251)

Cycle run

Use Cycle 251 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

Roughing

1. The tool plunges the workpiece at the pocket center and advances to the first plunging depth.
2. The TNC roughs out the pocket from the inside out, taking the path overlap (parameter Q370) and the finishing allowance (parameters Q368 and Q369) into account.
3. At the end of the roughing operation, the TNC moves the tool 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. If finishing allowances have been defined, the TNC plunges and then approaches the contour. The TNC first finishes the pocket walls, in multiple infeeds if so specified.
6. Then the TNC finishes the floor of the pocket from the inside out.

Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The TNC automatically pre-positions the tool in the tool axis. Observe **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.

At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed.

NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

NOTICE

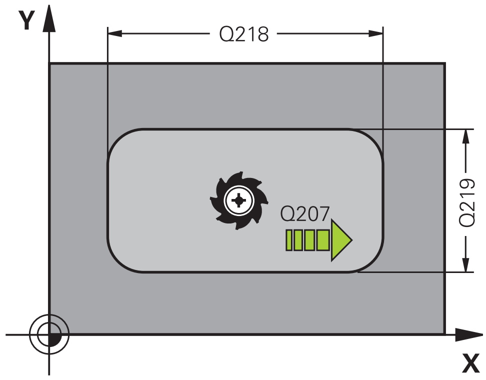
Danger of collision!

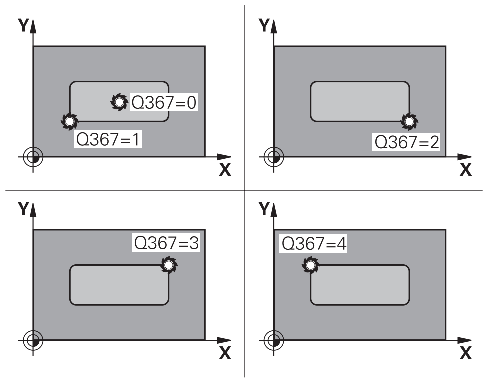
If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

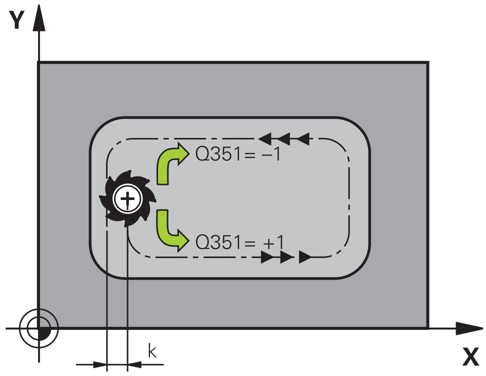
- Perform roughing beforehand
- Ensure that the TNC can pre-position the tool at rapid traverse without colliding with the workpiece

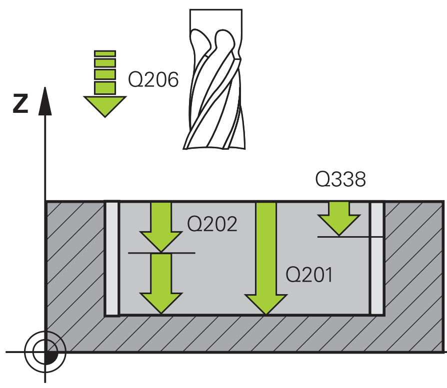
Cycle parameters

- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **Q218 First side length?** (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **Q219 Second side length?** (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Q367 Position of pocket (0/1/2/3/4)?**: Position of the pocket in reference to the position of the tool when the cycle is called:
  0: Tool position = pocket center
  1: Tool position = left corner below
  2: Tool position = right corner below
  3: Tool position = right corner top
  4: Tool position = left corner top
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **fu**, **FZ**
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**, **FZ**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO**, **fu**, **FZ**
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane. Input range 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999;

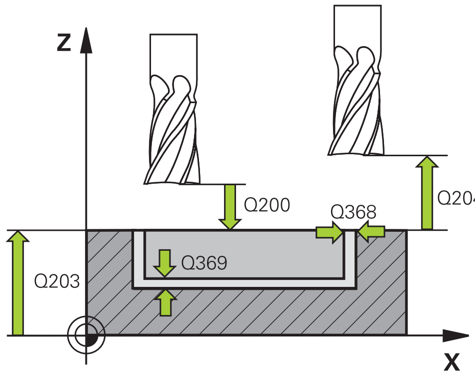








- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999;
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb milling
  –1 = Up-cut milling (if you enter 0, climb milling is performed)
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1.9999



NC blocks

8 CYCL DEF 251 RECTANGULAR POCKET	
Q215=0	;MACHINING OPERATION
Q218=80	;FIRST SIDE LENGTH
Q219=60	;2ND SIDE LENGTH
Q201=-20	;DEPTH
Q367=0	;POCKET POSITION
Q202=5	;PLUNGING DEPTH
Q207=500	;FEED RATE FOR MILLNG
Q206=150	;FEED RATE FOR PLNGNG
Q385=500	;FINISHING FEED RATE
Q368=0.2	;ALLOWANCE FOR SIDE
Q369=0.1	;ALLOWANCE FOR FLOOR
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q351=+1	;CLIMB OR UP-CUT
Q370=1	;TOOL PATH OVERLAP
9 X+50 R0 FMAX	
10 Y+50 R0 FMAX M3 M99	

18.3 SLOT MILLING (Cycle 253, DIN/ISO: G253)

Cycle run

Use Cycle 253 to completely machine a slot on a straight cut control. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, finishing
- Only roughing
- Only finishing

Roughing

1. The tool advances to the first PLUNGING DEPTH Q202 at the FEED RATE FOR PLUNGING Q206. The slot created by the roughing process is exactly as wide as the diameter of the tool. During roughing, the TNC only moves the tool in the tool axis and along the SLOT LENGTH Q218. If the SLOT WIDTH is greater than the tool diameter, a subsequent finishing operation needs to be programmed.
2. The TNC roughs out the slot, taking the parameters Q351 CLIMB OR UP-CUT and Q352 PLUNGING POSITION into account.
3. Depending on parameter Q352 PLUNGING POSITION, the downfeed is either reciprocating (bidirectional) or always from the same side (unidirectional).
   - Bidirectional: The tool performs a cut and then advances to the next plunging depth on the side on which the tool is currently located.
   - Unidirectional: The tool performs a cut, retracts by the set-up clearance Q200 and then returns to the starting position where it advances to the next plunging depth. The plunging motion is always performed on the same side.
4. This process is repeated until the programmed slot depth is reached.
5. Finally, the TNC retracts the tool to the set-up clearance Q200, moves it back to the center of the slot and then to the 2nd set-up clearance Q204.

Finishing

6. 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 left slot arc.
7. Then the TNC finishes the floor of the slot from the inside out.

Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The TNC automatically pre-positions the tool in the tool axis. Observe **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.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

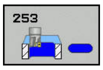
NOTICE

Danger of collision!

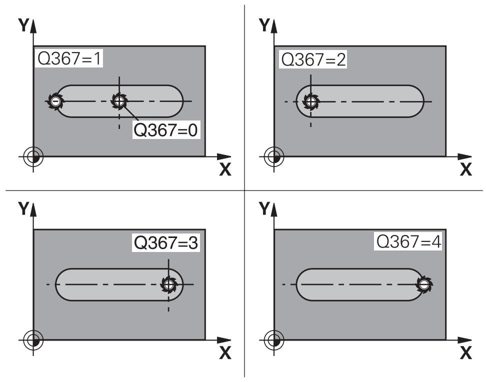
After roughing, the slot width equals the tool diameter, regardless of parameter Q219!

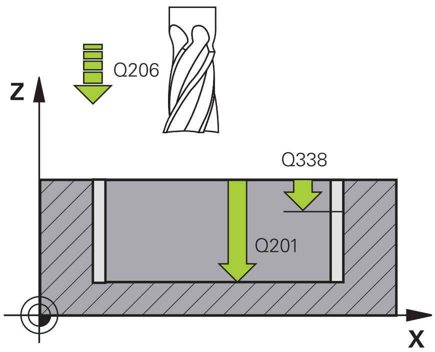
- If you use a small roughing tool, then a large amount of material can remain for the finishing tool; keep this in mind when selecting your tools!

Cycle parameters



- **Q215 Machining operation (0/1/2)?**: Define the extent of machining:
  0: Roughing and finishing
  1: Roughing only
  2: Finishing only
- **Q218 Length of slot?** (value parallel to the reference axis of the working plane): Enter the length of the slot. Input range 0 to 99999.9999
- **Q219 Width of slot?** (value parallel to the secondary axis of the working plane): Enter the slot width. After roughing, the slot is only as wide as the tool diameter, regardless of parameter Q219! Maximum slot width for finishing: Twice the tool diameter. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range -99999.9999 to 99999.9999
- **Q374 Slot direction?**: Enter whether the slot is rotated by 90° (vertical slot, input: 1) or whether it is not rotated (horizontal slot, input: 0). The center of rotation is at the center of the slot.
- **Q367 Position of slot (0/1/2/3/4)?**: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position = slot center
  1: Tool position = left end of slot
  2: Tool position = center of left slot arc
  3: Tool position = center of right slot arc
  4: Tool position = right end of slot
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **fu**, **FZ**
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**, **FZ**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range 0 to 99999.999, alternatively **FAUTO**, **fu**, **FZ**
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999;
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999





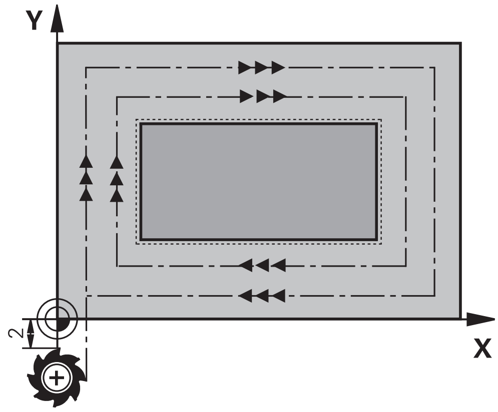
NC blocks

8 CYCL DEF 253 SLOT MILLING	
Q215=0	**;MACHINING OPERATION**
Q218=80	**;SLOT LENGTH**
Q219=12	**;SLOT WIDTH**
Q201=-20	**;DEPTH**
Q374=+0	**;SLOT DIRECTION**
Q367=0	**;SLOT POSITION**
Q202=5	**;PLUNGING DEPTH**
Q207=500	**;FEED RATE FOR MILLNG**
Q206=150	**;FEED RATE FOR PLNGNG**
Q385=500	**;FINISHING FEED RATE**
Q338=5	**;INFEED FOR FINISHING**
Q200=2	**;SET-UP CLEARANCE**
Q203=+0	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**
Q351=1	**;CLIMB OR UP-CUT**
Q352=0	**;PLUNGE POSITION**
9 L X+50 Y+50 R0 FMAX M3 M99	

- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999;
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The TNC uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q352 Plunge position?**: Specify at which position along the reference axis the tool is to plunge:
  +1: Plunging position always at the right end of the slot
  -1: Plunging position always at the left end of the slot
  0: Reciprocating plunge

18.4 RECTANGULAR STUD (Cycle 256)

Cycle run



Use Cycle 256 to machine a rectangular stud. If a dimension of the workpiece blank is greater than the maximum possible stepover, then the TNC performs multiple stepovers until the finished dimension has been machined.

1. The tool moves from the cycle starting position (stud center) in the negative X direction to the starting position for stud machining. The starting position is to the left of the unmachined stud and is offset by the set-up clearance + tool radius.
2. If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to the set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
3. The tool then moves on a straight line to the stud contour and machines one revolution.
4. If the finished dimension cannot be machined with one revolution, the TNC performs a stepover with the current factor, and machines another revolution. The TNC takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached.
5. If further stepovers are required the tool then departs the contour on and returns to the starting point of stud machining
6. The TNC then plunges the tool to the next plunging depth, and machines the stud at this depth.
7. This process is repeated until the programmed stud depth is reached.

Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The TNC automatically pre-positions the tool in the tool axis. Observe **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.

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

NOTICE

Danger of collision!

If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

NOTICE

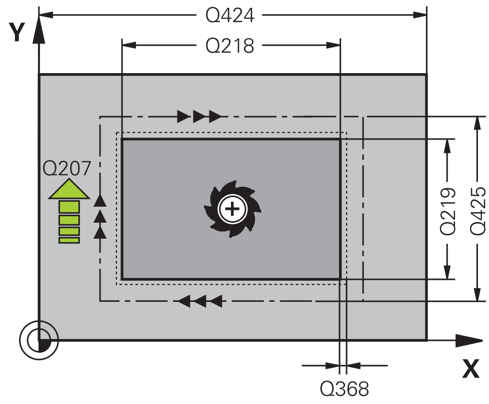
Danger of collision!

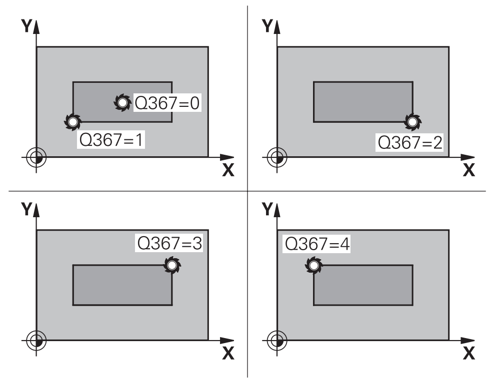
There is a danger of collision if there is insufficient room next to the stud.

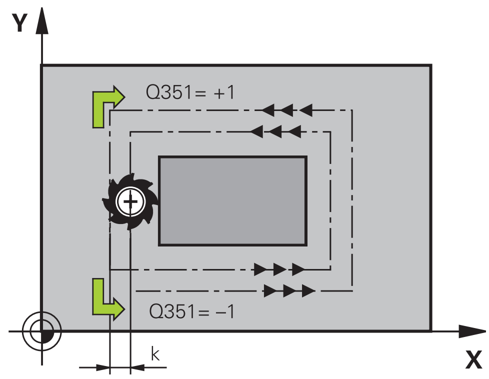
- Depending on the approach position Q439, the TNC requires room for the approach motion
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed. This means that the end position of the tool after the cycle differs from the starting position.

Cycle parameters

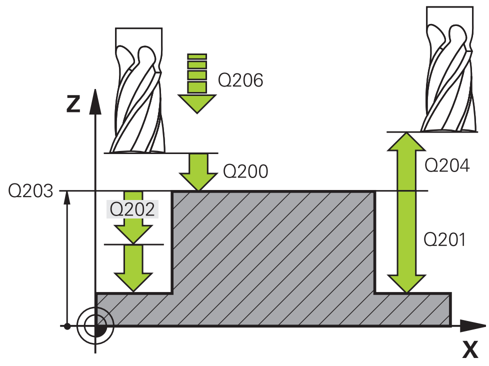
- **Q218 First side length?**: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **Q424 Workpiece blank side length 1?**: Length of the unmachined stud, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **First side length.** The TNC performs multiple stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **Q219 Second side length?**: Stud length, parallel to the minor axis of the working plane. Enter **Workpiece blank side length 2** greater than **Second side length** The TNC performs multiple stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- **Q425 Workpiece blank side length 2?**: Length of the unmachined stud, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- **Q367 Position of stud (0/1/2/3/4)?**: Position of the stud in reference to the position of the tool when the cycle is called:
  0: Tool position = stud center
  1: Tool position = left corner below
  2: Tool position = right corner below
  3: Tool position = right corner top
  4: Tool position = left corner top
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **fu**, **FZ**







- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min while moving to depth. Input range 0 to 99999.999; alternatively **fmax**, **FAUTO**, **fu**, **FZ**
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane, is left over after machining. Input range 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999;
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999;
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb milling
  –1 = Up-cut milling (if you enter 0, climb milling is performed)
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. The overlapping is considered as maximum overlapping. The overlapping can be reduced to avoid residual material at the corners. Input range 0.1 to 1.9999;



NC blocks

8 CYCL DEF 256 RECTANGULAR STUD	
Q215=0	;MACHINING OPERATION
Q218=60	;FIRST SIDE LENGTH
Q424=74	;WORKPC. BLANK SIDE 1
Q219=40	;2ND SIDE LENGTH
Q425=60	;WORKPC. BLANK SIDE 2
Q201=-20	;DEPTH
Q367=0	;STUD POSITION
Q202=5	;PLUNGING DEPTH
Q207=500	;FEED RATE FOR MILLNG
Q206=150	;FEED RATE FOR PLNGNG
Q385=500	;FINISHING FEED RATE
Q368=0.2	;ALLOWANCE FOR SIDE
Q369=0.1	;ALLOWANCE FOR FLOOR
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q351=+1	;CLIMB OR UP-CUT
Q370=1	;TOOL PATH OVERLAP
9 X+50 R0 FMAX	
10 Y+50 R0 FMAX M3 M99	

18.5 FACE MILLING (Cycle 233)

Cycle run

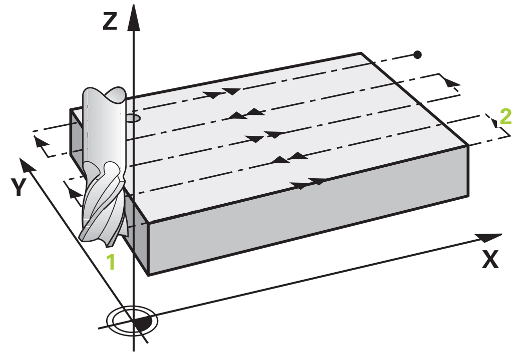
Cycle 233 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Additionally, you can also define side walls in the cycle which are taken into account during the machining of the level surface The cycle offers you various machining strategies:

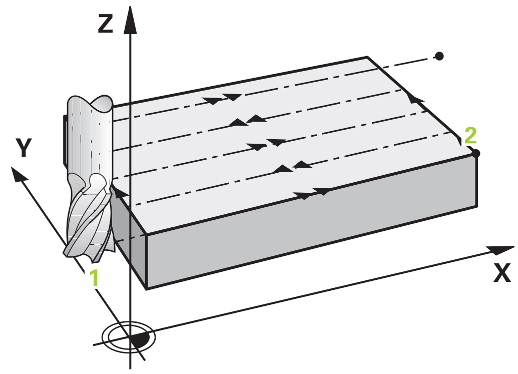
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover after retracting at rapid traverse
- **Strategy Q389=3**: The surface is machined line by line without overtravel; stepover after retracting at rapid traverse
- **Strategy Q389=4**: Helical machining from the outside toward the inside

1. From the current position, the TNC positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane: The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.
2. The TNC then positions the tool at rapid traverse **FMAX** to the set-up clearance in the spindle axis.
3. The tool then moves in the tool axis at the feed rate for milling Q207 to the first plunging depth calculated by the TNC.

Strategies Q389=0 and Q389 =1

The strategies Q389=0 and Q389=1 differ in the overtravel during face milling. If Q389=0, the end point lies outside of the surface. If Q389=1, it lies at the edge of the surface. The TNC calculates the end point **2** from the side length and the safety clearance to the side. If the strategy Q389=0 is used, the TNC additionally moves the tool beyond the level surface by the tool radius.

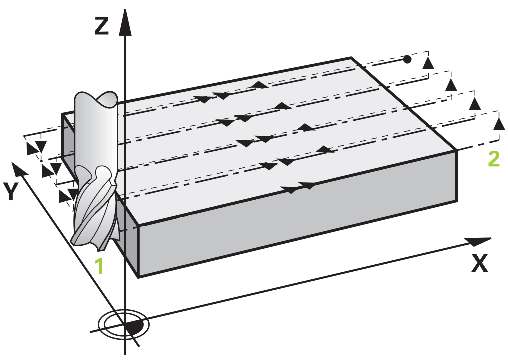


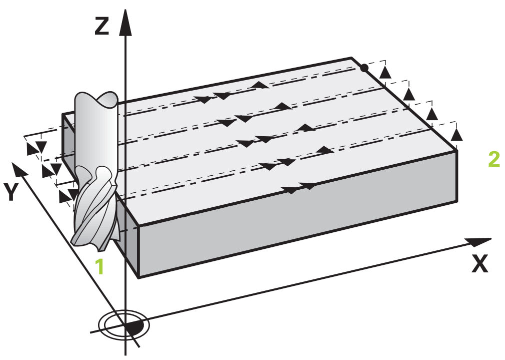


4 The TNC moves the tool to the end point **2** at the programmed feed rate for milling.
5 Then the TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and the safety clearance to the side.
6 The tool then returns at the feed rate for milling in the opposite direction.
7 The process is repeated until the programmed surface has been completed.
8 The TNC then positions the tool at rapid traverse **FMAX** back to the starting point **1**.
9 If more than one infeed is required, the TNC moves the tool in the tool axis to the next plunging depth at the positioning feed rate.
10 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.
11 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategies Q389=2 and Q389 =3

The strategies Q389=2 and Q389=3 differ in the overtravel during face milling. If Q389=2, the end point lies outside of the surface. If Q389=3, it lies at the edge of the surface. The TNC calculates the end point **2** from the side length and the safety clearance to the side. If the strategy Q389=2 is used, the TNC additionally moves the tool beyond the level surface by the tool radius.

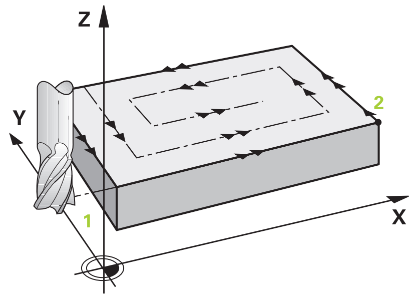




4 The tool subsequently advances to the end point **2** at the programmed feed rate for milling.
5 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at **FMAX**paraxially back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the safety clearance to the side.
6 The tool then returns to the current infeed depth and moves in the direction of the next end point **2**.
7 The multipass process is repeated until the programmed surface has been completed. At the end of the last path, the TNC positions the tool at rapid traverse **FMAX** back to the starting point **1**.
8 If more than one infeed is required, the TNC moves the tool in the tool axis to the next plunging depth at the positioning feed rate.
9 The process is repeated until all infeeds have been machined. In the last infeed, only the finishing allowance entered is milled at the finishing feed rate.
10 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

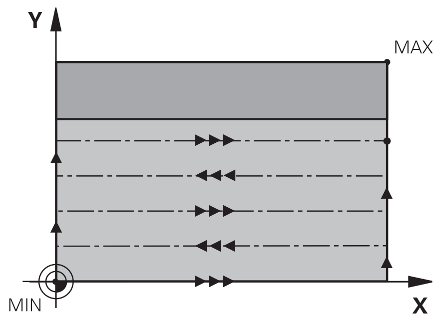
Strategy Q389=4

4 The tool subsequently approaches the starting point of the milling path at the programmed **Feed rate for milling** on a straight line tangential arc.
5 The TNC machines the level surface at the feed rate for milling from the outside toward the inside with ever-shorter milling paths. The constant stepover results in the tool being continuously engaged.
6 The process is repeated until the programmed surface has been completed. At the end of the last path, the TNC positions the tool at rapid traverse **FMAX** back to the starting point **1**.
7 If more than one infeed is required, the TNC moves the tool in the tool axis to the next plunging depth at the positioning feed rate.
8 The process is repeated until all infeeds have been machined. In the last infeed, only 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**.



Limits

The limiters enable you to limit the machining of the level surface, for example, to account for side walls or shoulders during machining. A side wall that is defined by a limit is machined to the finished dimension resulting from the starting point or the side lengths of the level surface. For roughing the TNC includes the oversize of the side - for finishing the oversize helps to preposition the tool.



Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Keep in mind the machining direction.

The TNC automatically pre-positions the tool in the tool axis. Observe **Q204 2ND SET-UP CLEARANCE**.

Enter the **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.

If **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS** are entered as equal values, the TNC will not carry out the cycle (depth=0 programmed).

The TNC reduces the infeed depth to the LCUTS tool length defined in the tool table if the tool length is shorter than the Q202 infeed depth programmed in the cycle.

If you define **Q370** TOOL PATH OVERLAP >1, the programmed overlap factor is taken into account from the first machining path.

Cycle 233 monitors the entry for tool length / length of the cutting edge **LCUTS** in the tool table. If the length of the tool or teeth is not sufficient for finishing operations, The TNC subdivides the machining into several working steps.

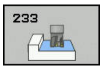
NOTICE

Danger of collision!

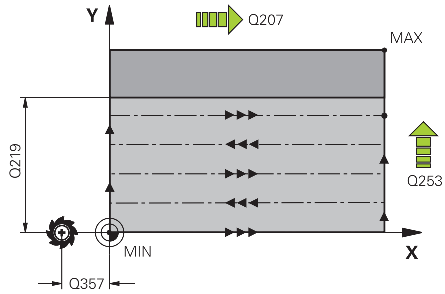
If you enter a positive depth with a cycle, the TNC reverses calculation of the pre-positioning. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

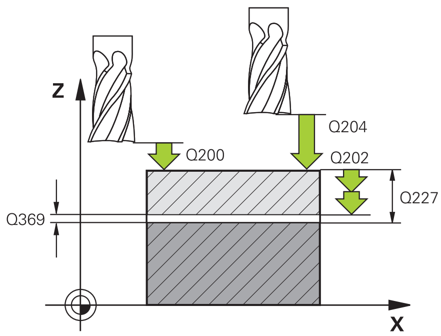
- Enter depth as negative
- Enter in machine parameter **displayDepthErr** (No. 201003) whether the TNC should output an error message (on) or not (off) if a positive depth is entered

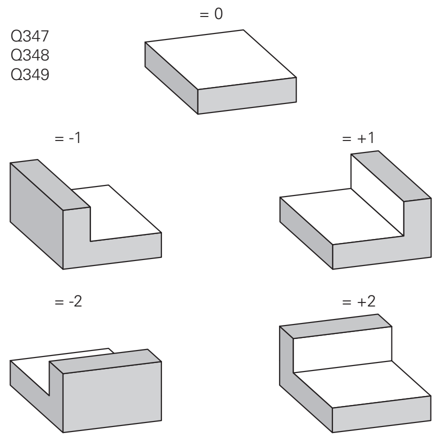
Cycle parameters



- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is defined
- **Q389 Machining strategy (0-4)?**: Determine how the TNC should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface being machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate
  3: Line-by-line machining, retraction and stepover at the edge of the surface being machined
  4: Helical machining, smooth approach from the outside toward the inside
- **Q350 Milling direction?**: Axis in the machining plane that defines the machining direction:
  1: Reference axis = machining direction
  2: Minor axis = machining direction
- **Q218 First side length?** (incremental): Length of the surface to be machined in the reference axis of the working plane, referenced to the starting point in the 1st axis. Input range -99999.9999 to 99999.9999
- **Q219 Second side length?** (incremental): 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 transverse approach in reference to the **STARTNG PNT 2ND AXIS**. Input range -99999.9999 to 99999.9999







- **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range -99999.9999 to 99999.9999
- **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis on which the surface is to be face-milled. Input range -99999.9999 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range 0 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range 0 to 99999.9999
- **Q370 Path overlap factor?**: 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. Input range: 0.1 to 1.9999.
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min while milling. Input range 0 to 99999.999 alternatively **FAUTO**, **fu**, **FZ**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range 0 to 99999.9999, alternatively **FAUTO**, **fu**, **FZ**
- **Q253 Feed rate for pre-positioning?**: 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. Input range 0 to 99999.9999, alternatively **fmax**, **FAUTO**
- **Q357 Safety clearance to the side?** (incremental) Parameter Q357 has an influence on the following situations:
  Approach the first plunging depth: Q357 is the safety clearance of the tool to the side of the workpiece
  Roughing with milling strategies Q389=0-3: The surface in **Q350** MILLING DIRECTION is increased by the value from Q357, provided that no limitation is set in this direction
  Side finishing: The paths are extended by Q357 in **Q350** MILLING DIRECTION
  Input range 0 to 99999.9999

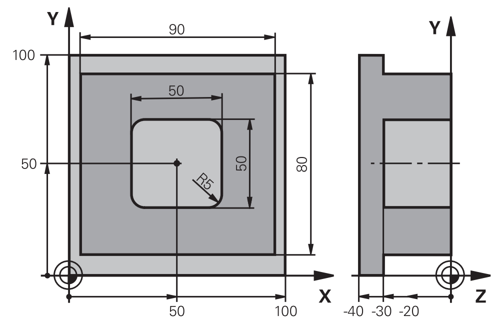
NC blocks

8 CYCL DEF 233 FACE MILLING	
Q215=0	;MACHINING OPERATION
Q389=2	;MILLING STRATEGY
Q350=1	;MILLING DIRECTION
Q218=120	;FIRST SIDE LENGTH
Q219=80	;2ND SIDE LENGTH
Q227=0	;STARTNG PNT 3RD AXIS
Q386=-6	;END POINT 3RD AXIS
Q369=0.2	;ALLOWANCE FOR FLOOR
Q202=3	;MAX. PLUNGING DEPTH
Q370=1	;TOOL PATH OVERLAP
Q207=500	;FEED RATE FOR MILLNG
Q385=500	;FINISHING FEED RATE
Q253=750	;F PRE-POSITIONING
Q357=2	;CLEARANCE TO SIDE
Q200=2	;SET-UP CLEARANCE
Q204=50	;2ND SET-UP CLEARANCE
Q347=0	;1ST LIMIT
Q348=0	;2ND LIMIT
Q349=0	;3RD LIMIT
Q368=0	;ALLOWANCE FOR SIDE
Q338=0	;INFEED FOR FINISHING
9 L X+0 Y+0 R0 FMAX M3 M99	

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999;
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999;
- **Q347 1st limit?**: Select the side of the workpiece where the plan surface is bordered by a side wall. Depending on the position of the side wall, the TNC limits the machining of the level surface to the respective coordinate of the starting point or to the side length: :
  Input **0**: No limiting
  Input **-1**: Limiting in negative principal axis
  Input **+1**: Limiting in positive principal axis
  Input **-2**: Limiting in negative secondary axis
  Input **+2**: Limiting in positive secondary axis
- **Q348 2nd limit?**: See Parameter 1st limit Q347
- **Q349 3rd limit?**: See Parameter 1st limit Q347
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the machining plane. Input range 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999

18.6 Programming Examples

Example: Milling pockets, studs



0 BEGINN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Call the tool for roughing/finishing
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD	Define cycle for machining the contour outside
Q218=90 ;FIRST SIDE LENGTH	
Q424=100 ;WORKPC. BLANK SIDE 1	
Q219=80 ;2ND SIDE LENGTH	
Q425=100 ;WORKPC. BLANK SIDE 2	
Q201=-30 ;DEPTH	
Q367=0 ;STUD POSITION	
Q202=5 ;PLUNGING DEPTH	
Q207=250 ;FEED RATE FOR MILLNG	
Q206=250 ;FEED RATE FOR PLNGNG	
Q385=750 ;FINISHING FEED RATE	
Q368=0 ;ALLOWANCE FOR SIDE	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q351=+1 ;CLIMB OR UP-CUT	
Q370=1 ;TOOL PATH OVERLAP	
6 X+50 R0	Outside machining
7 Y+50 R0 M3 M99	Call cycle for machining the contour outside
8 CYCL DEF 252 RECTANGULAR POCKET	Define RECTANGULAR POCKET cycle
Q215=0 ;MACHINING OPERATION	
Q218=50 ;FIRST SIDE LENGTH	
Q219=50 ;2ND SIDE LENGTH	

Q201=-30 ;DEPTH	
Q367=+0 ;POCKET POSITION	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE FOR MILLNG	
Q206=150 ;FEED RATE FOR PLNGNG	
Q385=750 ;FINISHING FEED RATE	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q351=+1 ;CLIMB OR UP-CUT	
Q370=1 ;TOOL PATH OVERLAP	
9 X+50 R0 FMAX	
10 Y+50 R0 FMAX M99	Cycle call
11 Z+250 R0 FMAX M30	
12 END PGM C210 MM	

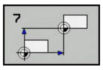
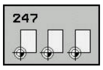
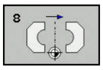
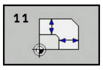
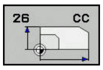
19

Cycles: Coordinate Transformations

19.1 Fundamentals

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:

Soft key	Cycle	Page
	7 DATUM SHIFT For shifting contours directly within the program or from datum tables	595
	247 PRESETTING Presetting during program run	601
	8 MIRRORING Mirroring contours	602
	11 SCALING FACTOR Increasing or reducing the size of contours	603
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	604

Effectiveness of coordinate transformations

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

Reset coordinate transformation:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM block (depending on machine parameter **clearMode**)
- Select a new program

19.2 DATUM SHIFT (Cycle 7)

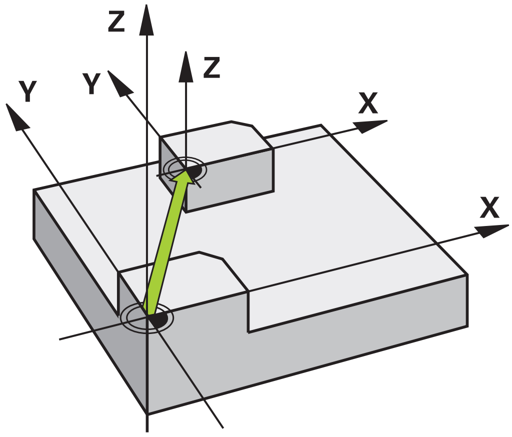
Effect

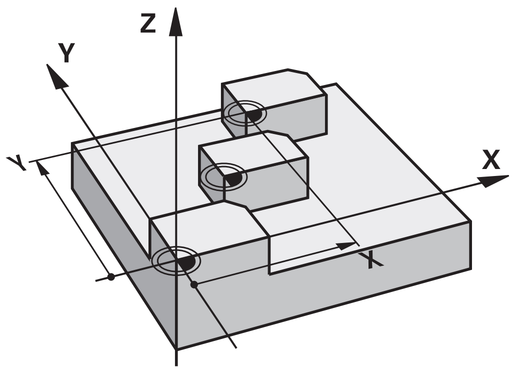
A datum shift allows machining operations to be repeated at various locations on the workpiece.

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.

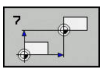
Resetting

- Program a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.





Cycle parameters



- **Displacement**: Enter the coordinates of the new datum. Absolute values are referenced to the workpiece datum, which is specified by presetting. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted. Input range: Up to six NC axes, each from –99999.9999 to 99999.9999

NC blocks

13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 7.3 Z-5

Please note while programming

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

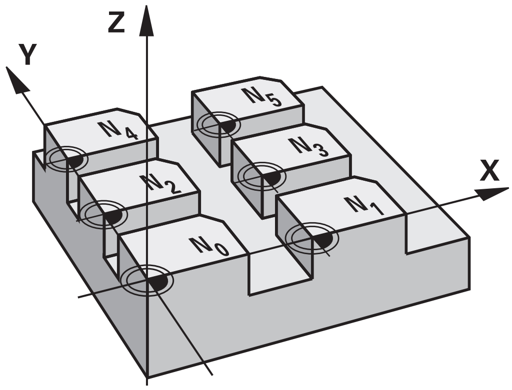
19.3 DATUM SHIFT with datum tables (Cycle 7)

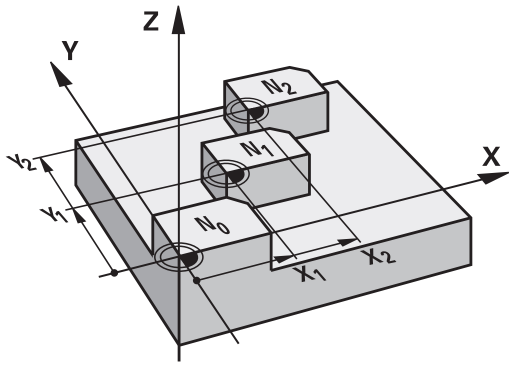
Effect

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 datums directly in the cycle definition or call them from a datum table.





Resetting

- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.
- Execute a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition

Status displays

In the additional status display, the following data from the datum table are shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

Please note while programming:

Datums from a datum table are **always and exclusively** referenced to the current preset.

If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

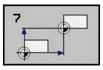
If you work without **SEL TABLE,** then you must activate the desired datum table before the test run or the program run. (This applies also to the programming graphics).

- 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 to select the desired table for the program run in the **Program run, single block** and **Program run, full sequence** operating modes: The table receives the status M

The coordinate values from datum tables are only effective with absolute coordinate values.

If you create datum tables, the file name has to start with a letter.

Cycle parameters



- **Displacement**: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number entered in the Q parameter. Input range 0 to 9999

NC blocks

```
77 CYCL DEF 7.0 DATUM SHIFT
78 CYCL DEF 7.1  #5
```

Selecting a datum table in the part program

With the **SEL TABLE** function you select the table from which the TNC takes the datums:

PGM CALL

- To select the functions for program call, press the **PGM CALL** key

DATUM TABLE

- Press the **DATUM TABLE** soft key
- Select the complete path name of the datum table or the file with the **SELECT** soft key and confirm your entry with the **END** key

> Program a **SEL TABLE** block before Cycle 7 Datum Shift.
>
> A datum table selected with **SEL TABLE** remains active until you select another datum table with **SEL TABLE** or through **PGM MGT**.

Editing the datum table in the Programming mode of operation

> After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise the change might not be included during program run.

Select the datum table in the mode of operation **Programming**

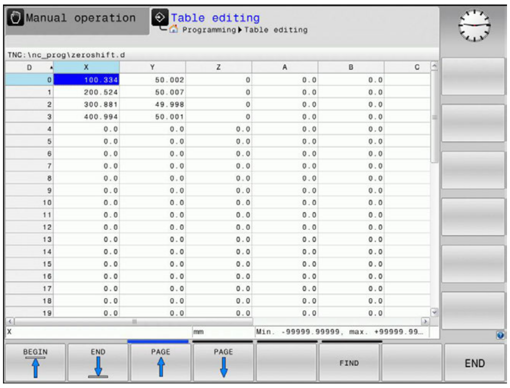
PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Display the datum tables: Press the **SELECT TYPE** and **SHOW .D** soft keys
- Select the desired table or enter a new file name.
- Edit the file. The functions displayed in the soft-key row for editing include:

Soft key	Function
BEGIN	Select beginning of table
END	Select the table end
PAGE	Go to previous page
PAGE	Go to next page
INSERT LINE	Insert line
DELETE LINE	Delete line
FIND	Find
BEGIN LINE	Go to beginning of line
END LINE	Go to end of line
COPY FIELD	Copy the current value
PASTE FIELD	Insert the copied value
APPEND N LINES	Add the entered number of lines (datums) to the end of the table

Configuring a datum table

If you do not wish to define a datum for an active axis, press the **CE** key. Then the TNC clears the numerical value from the corresponding input field.



> You can change the properties of tables. Enter the code number 555343 in the MOD menu. The TNC then offers the **EDIT FORMAT** soft key if a table is selected. When you press this soft key, the TNC opens a pop-up window where the properties are shown for each column of the selected table. Any changes made only affect the open table.

Leaving a datum table

Select a different type of file in file management and choose the desired file.

NOTICE

Danger of collision!

The control considers changes in a datum table only when the values are saved.

- Confirm changes in the table immediately with the ENT key
- Carefully test the NC program after making a change to the datum table

Status displays

In the additional status display, the TNC shows the values of the active datum shift.

19.4 PRESETTING (Cycle 247)

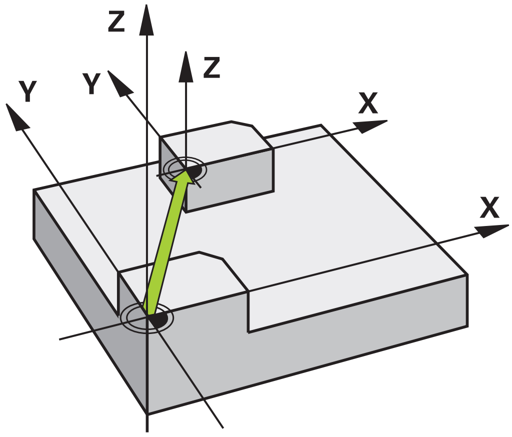
Effect

With the presetting cycle you can activate as the new preset a preset defined in the preset table.

After a presetting cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

Status display

In the status display the TNC shows the active preset number behind the preset symbol.



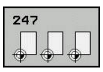
Please note before programming:

When activating a preset from the preset table, the TNC resets the datum shift, mirroring, scaling factor and axis-specific scaling factor.

If you activate preset number 0 (row 0), then you activate the preset that you last set in **Manual operation** or **Electronic handwheel** operating mode.

Cycle 247 is also effective in the Test Run mode of operation.

Cycle parameters



- **Number for preset?**: Enter the number of the desired preset from the preset table. As an alternative, you can also select the desired preset directly from the preset table with **SELECT**. Input range 0 to 65535

NC blocks

```
13 CYCL DEF 247 PRESETTING
    Q339=4      ;PRESET NUMBER
```

19.5 MIRRORING (Cycle 8)

Effect

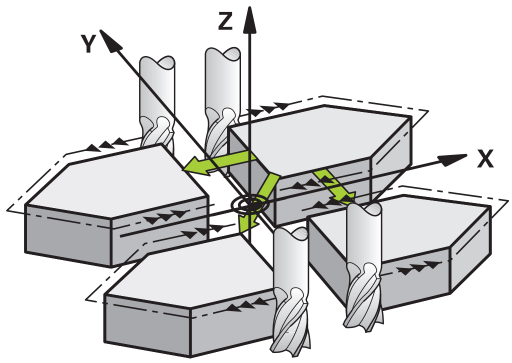
The TNC can machine the mirror image of a contour in the working plane.

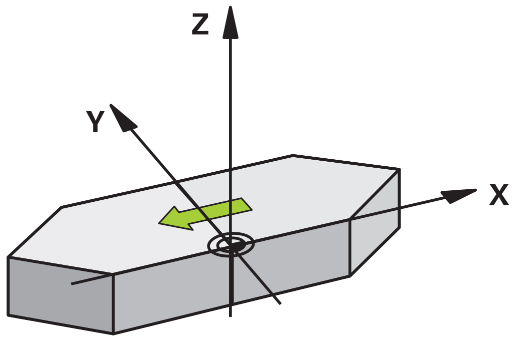
The mirroring cycle becomes effective as soon as it is defined in the program. It is also effective in **Positioning w/ Manual Data Input** mode of operation. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed
- If you mirror two axes, the machining direction remains the same.

The result of the mirroring 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.

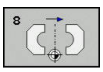




Resetting

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

Cycle parameters



- **Mirror image 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. Input range: Up to three NC axes **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

NC blocks

```
79 CYCL DEF 8.0  MIRRORING
80 CYCL DEF 8.1  X Y  Z
```

19.6 SCALING (Cycle 11

Effect

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in **Positioning w/ Manual Data Input** mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- all three coordinate axes at the same time
- dimensions in cycles

Prerequisite

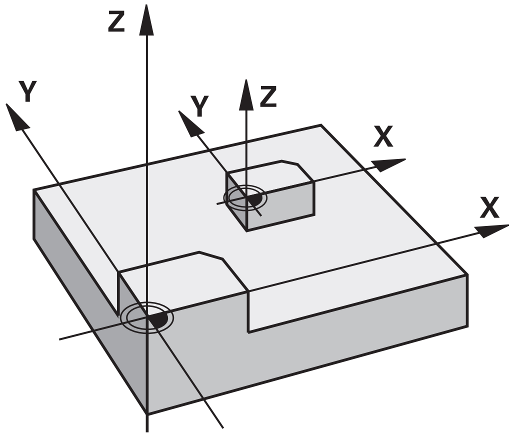
It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

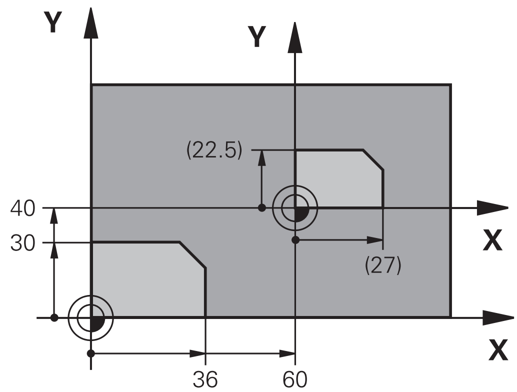
Enlargement: SCL greater than 1 (up to 99.999 999)

Reduction: SCL less than 1 (down to 0.000 001)

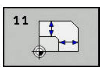
Resetting

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





Cycle parameters



- **Factor?**: Enter the scaling factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range 0.000001 to 99.999999

NC blocks

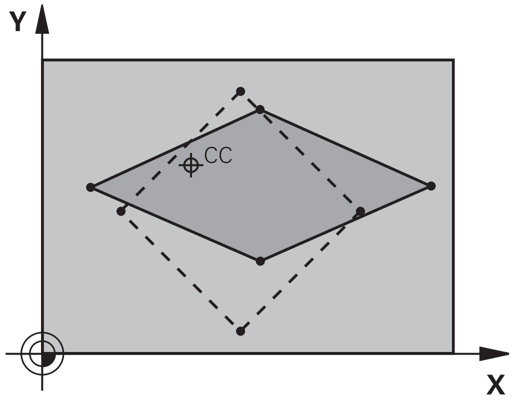
11 CALL LBL 1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1

19.7 AXIS-SPECIFIC SCALING (Cycle 26)

Effect

With Cycle 26 you can account for shrinkage and oversize factors for each axis.

The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in **Positioning w/ Manual Data Input** mode of operation. The active scaling factor is shown in the additional status display.



Resetting

Program the SCALING cycle once again with a scaling factor of 1 for the same axis.

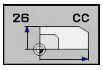
Please note while programming:

You can program each coordinate axis with its own axis-specific scaling factor.

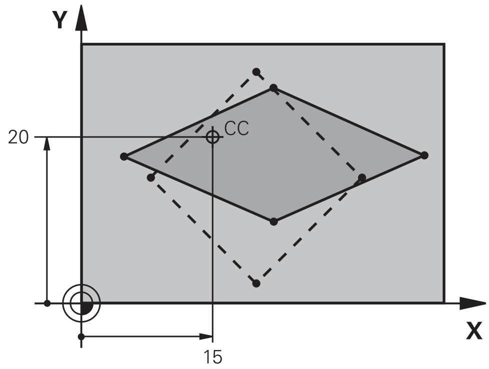
In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.

Cycle parameters



- **Axis and scaling factor**: Select the coordinate axis/axes by soft key and enter the factor(s) involved in enlarging or reducing. Input range 0.000001 to 99.999999
- **Center coordinates**: Enter the center of the axis-specific enlargement or reduction. Input range -99999.9999 to 99999.9999



NC blocks

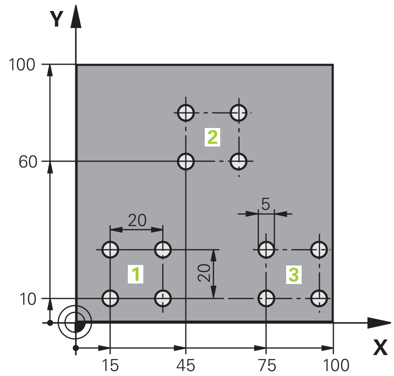
25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL 1

19.8 Programming Examples

Example: Groups of holes

Program run:

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1) in the main program
- Program the group of holes only once in subprogram 1



0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3000	Tool call
4 Z+250 R0 FMAX M3	
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q210=+0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DWELL TIME AT DEPTH	
Q395=+0 ;DEPTH REFERENCE	
6 CYCL DEF 7.0 DATUM SHIFT	Datum shift
7 CYCL DEF 7.1 X+15	
8 CYCL DEF 7.2 Y+10	
9 CALL LBL 1	
10 CYCL DEF 7.0 DATUM SHIFT	Datum shift
11 CYCL DEF 7.1 X+75	
12 CYCL DEF 7.2 Y+10	
13 CALL LBL 1	
14 CYCL DEF 7.0 DATUM SHIFT	Datum shift
15 CYCL DEF 7.1 X+45	
16 CYCL DEF 7.2 Y+60	
17 CALL LBL 1	
18 CYCL DEF 7.0 DATUM SHIFT	
19 CYCL DEF 7.1 X+0	

20 CYCL DEF 7.2 Y+0	
21 Z+100 R0 FMAX M30	
22 LBL 1	
23 X+0 R0 FMAX	
24 Y+0 R0 FMAX M99	Move to 1st hole, call cycle
25 X+20 R0 FMAX M99	Move to 2nd hole, call cycle
26 Y+20 R0 FMAX M99	Move to 3rd hole, call cycle
27 X-20 R0 FMAX M99	Move to 4th hole, call cycle
28 LBL 0	
29 END PGM SP2 MM	

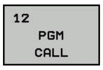
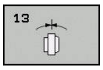
20

Cycles: Special Functions

20.1 Fundamentals

Overview

The TNC provides the following cycles for the following special purposes:

Soft key	Cycle	Page
	9 DWELL TIME	611
	12 Program call	612
	13 Oriented spindle stop	613

20.2 DWELL TIME (Cycle 9)

Function

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.

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

NC blocks

```
89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5
```

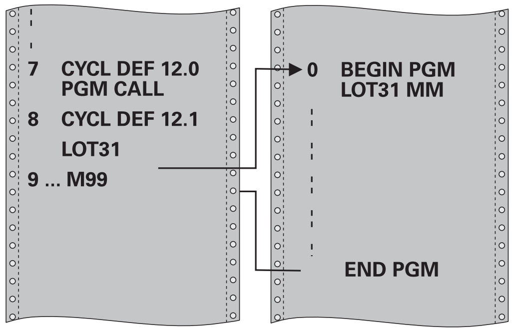
Cycle parameters

- **Dwell time in seconds**: Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

20.3 PROGRAM CALL (Cycle 12)

Cycle function

Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs. These can then be called like fixed cycles.



Please note while programming:

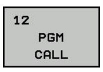
The program you are calling must be stored in the internal memory of your TNC.

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 enter the program name.

If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.

As a rule, Q parameters are globally effective when called with Cycle 12. So please note that changes to Q parameters in the called program can also influence the calling program.

Cycle parameters



- **Program name**: Enter the name of the program you want to call and, if necessary, the directory it is located in or
- Activate the file select dialog and select the program to be called via the **SELECT** soft key.

Call the program with:

- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Designate program 50 as a cycle and call it with M99

55 CYCL DEF 12.0 PGM CALL
56 CYCL DEF 12.1 PGM TNC:\KLAR35\FK1\50.H
57 X+20 FMAX
58 Y+50 FMAX M99

20.4 SPINDLE ORIENTATION (Cycle 13)

Cycle function

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

The TNC can control the machine tool spindle and rotate it to a given angular position.

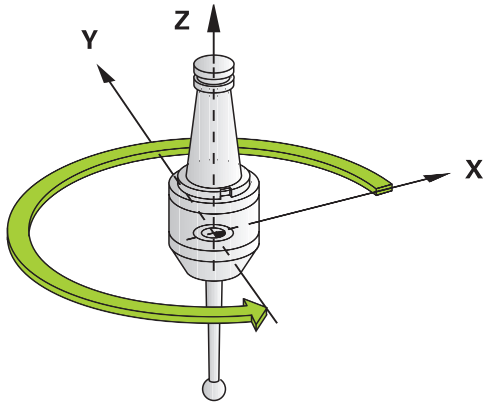
Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 13, the TNC positions the machine tool spindle at an angle that has been set by the machine tool builder.

More information: machine tool manual.



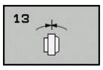
NC blocks

```
93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180
```

Please note while programming:

> Cycle 13 is used internally for Cycles 202 and 204. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

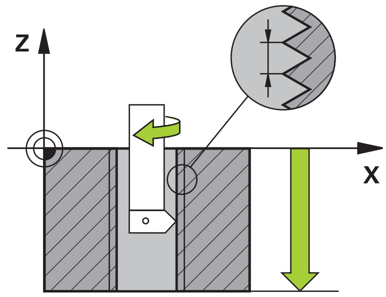


- **Angle of orientation**: Enter the angle referenced to the reference axis of the working plane. Input range: 0.0000° to 360.0000°

20.5 THREAD CUTTING (Cycle 18)

Cycle run

Cycle **18** THREAD CUTTING moves the tool with servo-controlled spindle from the momentary position with active speed to the entered depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.



Please note while programming:

It is possible to use the feed rate potentiometer during tapping. The machine tool builder sets the configuration (with parameter **CfgThreadSpindle>sourceOverride**) for this purpose. The TNC then modifies the speed accordingly.

The spindle speed potentiometer is inactive.

Program a spindle stop before starting the cycle! (e.g. with M5). The TNC switches the spindle on automatically at cycle start and off at the end of the cycle.

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

NOTICE

Danger of collision!

A collision may occur if you do not program pre-positioning before calling Cycle 18. Cycle 18 does not perform approach and departure motion.

- Before calling the cycle, pre-position the tool
- The tool moves from the current position to the entered depth after the cycle is called

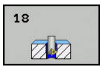
NOTICE

Danger of collision!

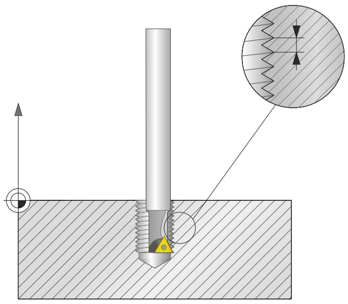
If the spindle was switched on before calling the cycle, Cycle 18 switches the spindle off and the cycle works with a stationary spindle! Cycle 18 switches the spindle on again at the end if it was switched on before cycle start.

- Program a spindle stop before starting the cycle! (e.g. with M5)
- After Cycle 18 has finished the spindle condition before cycle start is restored. If the spindle was switched off before cycle start, the TNC switches it off again at the end of Cycle 18.

Cycle parameters



- Boring depth (incremental): Enter the thread depth based on the current position Input range: -99999 to +99999
- Thread pitch: Enter the pitch of the thread. The algebraic sign entered here differentiates between right-hand and left-hand threads:
  + = right-hand thread (M3 with negative hole depth)
  - = left-hand thread (M4 with negative hole depth)



NC blocks

```
25 CYCL DEF 18.0 THREAD CUTTING
26 CYCL DEF 18.1 DEPTH = -20
27 CYCL DEF 18.2 PITCH = +1
```

21

Touch probe cycles

21.1 General information about touch probe cycles

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

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

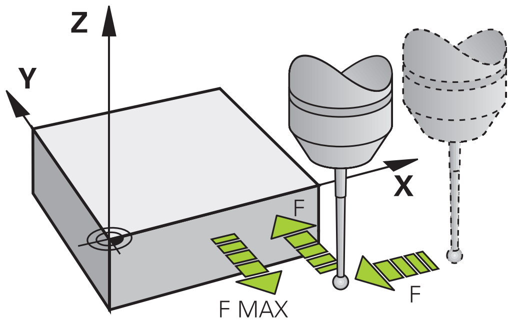
Touch-probe functions are not possible in combination with the **Global Program Settings** function. If at least one settings possibility is active, the control displays an error message if a manual touch-probe function is selected or when executing an automatic touch-probe cycle.

The touch probe cycles are available only with option 17. If you are using a HEIDENHAIN touch probe, this option is automatically available.

Method of function

Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Further information: "Before You Start Working with Touch Probe Cycles", page 619



When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the TNC: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the TNC displays an error message (distance: **DIST** from touch probe table).

Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes

In the **Manual operation** and **Electronic handwheel** modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Setting presets

The manual touch probe cycles are described in the "Manual operation and setup".

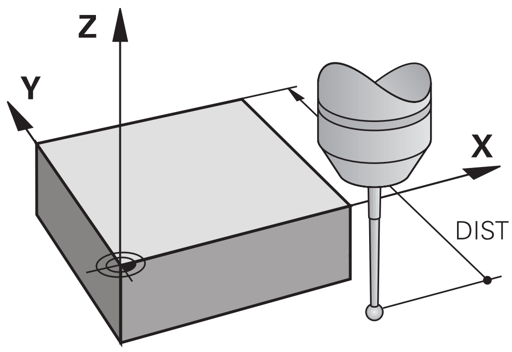
Further information: "Using a 3-D touch probe (option 17)", page 405

21.2 Before You Start Working with Touch Probe Cycles

To make it possible to cover the widest possible range of applications, machine parameters enable you to determine the behavior common to all touch probe cycles.

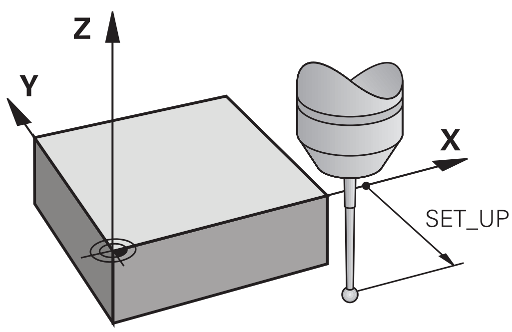
Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the path defined in **DIST**, the TNC outputs an error message.



Set-up clearance to touch point: SET_UP in touch probe table

In **SET_UP** you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to **SET_UP**.



Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table

To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

> If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch trigger probe, probing feed rate: F in touch probe table

In **F** you define the feed rate at which the TNC is to probe the workpiece.

F can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).

The feed rate potentiometer may be effective with touch probe cycles. The machine tool builder defines the required settings. (the parameter **overrideForMeasure** (No. 122604) must be appropriately configured.)

Touch trigger probe, rapid traverse for positioning: FMAX

In **FMAX** you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table

In **F_PREPOS** you define whether the TNC is to position the touch probe at the feed rate defined in FMAX or at rapid traverse.

- Input value = **FMAX_PROBE**: Position at feed rate from **FMAX**
- Input value = **FMAX_MACHINE**: Pre-position at rapid traverse

Executing touch probe cycles

All touch probe cycles are DEF active. This means that the TNC runs the cycle automatically as soon as the TNC executes the cycle definition in the program run.

NOTICE

Danger of collision!

Cycles for coordinate transformation must not be active during execution of the touch probe cycles.

- Do not activate the following cycles before using touch probe cycles: **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, **10 ROTATION**,Cycle **11 SCALING** and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

Touch probe cycles with a number greater than 400 position the touch probe according to a positioning logic:

- If the current coordinate of the stylus south pole is less than the coordinate of the clearance height (as defined in the cycle), then the TNC first retracts the touch probe in the touch probe axis to the clearance height and then positions it in the working plane near the first touch point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the TNC first positions the touch probe to the first probe point in the working plane, and then in the touch-probe axis directly to the measuring height

21.3 Touch probe table

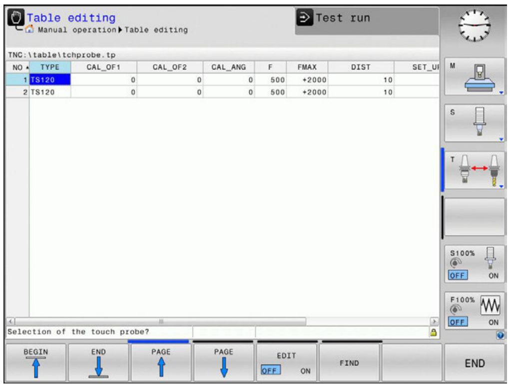
General information

Various data is stored in the touch probe table that defines the probe behavior during the probing process. If you use several touch probes on your machine tool, you can save separate data for each touch probe.

You can also view and edit the data of the touch probe table in the expanded tool management (option 93).

Editing touch probe tables

To edit the touch probe table, proceed as follows:



- Mode of operation: Press the **Manual operation** key

- Select the touch probe functions: Press the **TOUCH PROBE** soft key. The TNC displays additional soft keys

- Select the touch probe table: Press the **TCH PROBE TABLE** soft key

- Set the **EDIT** soft key to **ON**
- Using the arrow keys, select the desired setting.
- Perform desired changes.
- Exit the touch probe table: Press the **END** soft key

touch probe data

Abbr.	Inputs	Dialog
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool number	–
TYPE	Selection of the touch probe used	**Selection of the touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis in the principal axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis in the minor axis	**TS center misalignmt. aux. axis? [mm]**
CAL_ANG	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the control will probe the workpiece **F** can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe is pre-positioning and is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within this defined value, the control will issue an error message.	**Maximum measuring range? [mm]**
SET_UP	In **SET-UP** you define how far from the defined or calculated touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to the **SET-UP** machine parameter.	**Set-up clearance? [mm]**
F_PREPOS	Defining speed with pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-position at rapid? ENT/NOENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Probe oriented? Yes=ENT/No=NOENT**
SERIAL	You need not make an entry in this column. The TNC automatically enters the serial number of the touch probe if the touch probe has an EnDat interface	

21.4 Fundamentals

Overview

Operating notes

- When running touch probe cycles, Cycle **8 MIRROR IMAGE**, Cycle **11 SCALING** and Cycle **26 AXIS-SPECIFIC SCALING** must not be active.
- HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.

The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe.

Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.

The touch probe cycles are available only with the Touch Probe Functions software option (option 17). If you are using a HEIDENHAIN touch probe, this option is available automatically.

In conjunction with the TNC's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Tool measurement while the tool is at standstill
- Tool measurement while the tool is rotating
- Measurement of individual teeth

You can program the cycles for tool measurement in the **Programming** mode of operation using the **CYCL DEF** key. The following cycles are available:

New format	Cycle	Page
	Calibrating the TT, Cycle 480	628
	Measuring the tool length, Cycle 481	632
	Measuring the tool radius, Cycle 482	634
	Measuring the tool length and radius, Cycle 483	636

The measuring cycles can be used only when the central tool file TOOL.T is active.

Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.

Setting machine parameters

> Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (No. 122700) and **CfgTTRoundStylus** (No. 114200).
>
> Touch probe cycles 480, 481, 482, 483 and 484 can be hidden with the machine parameter **hideMeasureTT** (No. 128901).
>
> The TNC uses the feed rate for probing defined in the machine parameter **probingFeed** (No.122709) when measuring a tool at standstill.

When measuring a rotating tool, the TNC automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius in mm

The feed rate for probing is calculated from:
v = measuring tolerance • n with

v:	Feed rate for probing in mm/min
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc (No. 122710) determines the calculation of the probing feed rate:

probingFeedCalc (No. 122710) = **ConstantTolerance**:

The measuring tolerance remains constant regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The smaller you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** No. 122712) and the permissible tolerance (**measureTolerance1** No. 122715), the sooner you will encounter this effect.

probingFeedCalc (No. 122710) = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The TNC adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc (No. 122710) = **ConstantFeed**:

The feed rate for probing remains constant; the error of measurement, however, rises linearly with the increase in tool radius:

Measuring tolerance = r • **measureTolerance1**/ 5 mm, where

r:	Active tool radius in mm
measureTolerance1:	Maximum permissible error of measurement

Entries in the tool table TOOL.T

Abbr.	Inputs	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 **I**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the TNC locks the tool (status **I**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT.	Cutting direction of the tool for measuring the tool during rotation	**Cutting direction (M3 = -)?**
R_OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L_OFFS	Tool radius measurement: tool offset in addition to **offsetToolAxis** 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 **I**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	TT:R_OFFS	TT:L_OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill with diameter of < 19 mm	4 (4 teeth)	0 (no offset required because tool diameter is smaller than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from **offsetToolAxis** is used)
End mill with diameter of > 19 mm	4 (4 teeth)	R (offset required because tool diameter is larger than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from **offsetToolAxis** is used)
Radius cutter with a diameter of 10 mm, for example	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured)	5 (always define the tool radius as the offset so that the diameter is not measured in the radius)

21.5 Calibrating the TT (Cycle 480, option 17)

Cycle run

The TT is calibrated with the measuring cycle TCH PROBE 480. . The calibration process runs automatically. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.

Calibration process:

1. Clamp the calibrating tool. The calibrating tool must be a precisely cylindrical part, for example a cylinder pin
2. Manually position the calibrating tool in the working plane via the center of the TT
3. Position the calibrating tool in the tool axis approx. 15 mm + safety clearance above the TT
4. The TNC first moves along the tool axis. The tool is first moved to a clearance height of 15 mm + safety clearance
5. The calibration process along the tool axis starts
6. Calibration then follows in the working plane
7. The TNC initially positions the calibrating tool in the working plane at a value of 11 mm + radius of TT + safety clearance
8. Then the TNC moves the tool downwards along the tool axis and the calibration process starts
9. During the probing the TNC performs a square movement pattern
10. The TNC saves the calibration values and considers these during subsequent tool measurement
11. The TNC then retracts the stylus along the tool axis to safety clearance and moves it to the center of the TT

Please note while programming:

The functioning of the calibration cycle is dependent on machine parameter **CfgTTRoundStylus** (No. 114200). Refer to your machine manual.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The position of the TT within the machine working space must be defined by setting the machine parameters **centerPos** (No. 114201) > **[0]** to **[2]**.

If you change the setting of any of the machine parameters **centerPos** (No. 114201) > **[0]** to **[2]**, you must recalibrate.

Cycle parameters

- **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistToolAx**). Input range -99999.9999 to 99999.9999

NC blocks in new format

```
6 TOOL CALL 1 Z
7 TCH PROBE 480 CALIBRATE TT
    Q260=+100 ;CLEARANCE HEIGHT
```

21.6 Calibrating the wireless TT 449 (Cycle 484, Option 17)

Fundamentals

With Cycle 484, you can calibrate your tool touch probe, e.g the wireless infrared TT 449 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle 484 you must move the tool over the TT

Cycle run

To calibrate the tool touch probe, program the measuring cycle TCH PROBE 484. In the input parameter Q536, you can specify whether you want to run the cycle semi-automatically or fully automatically.

Semi-automatic—stop before running

- ▶ Insert the calibrating tool
- ▶ Define and start the calibration cycle
- ▶ The TNC interrupts the calibration cycle
- ▶ The TNC opens a dialog in a new window
- ▶ The dialog asks you to manually position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact

Fully automatic—no stop before running

- ▶ Insert the calibrating tool
- ▶ Position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact
- ▶ Define and start the calibration cycle
- ▶ The calibration cycle is executed without stopping. The calibration process starts from the current position of the tool.

Calibrating tool:

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. Enter the exact length and radius of the calibrating tool into the tool table TOOL.T. After the calibration, the TNC stores the calibration values and takes them into account during subsequent tool measurements. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Please note while programming:

NOTICE

Danger of collision!

To avoid collisions the tool must be pre-positioned before calling the cycle with Q536=1! In the calibration process, the TNC also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

- Specify whether to stop before cycle start or run the cycle automatically without stopping.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. When using a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. The use of a calibrating tool of too small a diameter and/or protruding too far from the chuck may cause significant inaccuracies.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

Q536 Stop before running (0=Stop)?: Specify whether to stop before cycle start or run the cycle automatically without stopping:
0: Stop before running. A dialog asks you to manually position the tool above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC start to continue the calibration process or press the **CANCEL** soft key to cancel the calibration process
1: No stop before running. The TNC starts the calibration process from the current position. Before running Cycle 484, you must position the tool above the tool touch probe.

21.7 Measuring tool length (Cycle 481, Option 17)

Cycle run

To measure the tool length, program the measuring cycle TCH PROBE 481. Via input parameters you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is at standstill.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is at standstill.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**TT: R_OFFS**).

Cycle for measuring a tool during standstill (e.g. for drills)

The control positions the tool to be measured over the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. To activate this function, enter zero for the tool offset: Radius (**TT: R_OFFS**) in the tool table.

Cycle for measuring individual teeth

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. You can enter an additional offset with tool offset: Length (**TT: L_OFFS**) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation.

Please note while programming:

> Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
>
> You can run an individual tooth measurement of tools with **up to 20 teeth**.

Cycle parameters

- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table
  0: The measured tool length is written to column L of the tool table TOOL.T, and the tool compensation is set to DL=0. If there is already a value stored in TOOL.T, it will be overwritten.
  1: The measured tool length is compared to the tool length L from TOOL.T. It then calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T)
  2: The measured tool length is compared to the tool length L from TOOL.T. The TNC calculates the deviation from the stored value and enters it in Q parameter Q115. Nothing is entered under L or DL in the tool table.
- **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth)

NC blocks

```
6 TOOL CALL 12 Z
7 TCH PROBE 481 CAL. TOOL LENGTH
    Q340=1      ;CHECK
    Q260=+100   ;CLEARANCE HEIGHT
    Q341=1      ;PROBING THE TEETH
```

21.8 Measuring tool radius (Cycle 482, Option 17)

Cycle run

To measure the tool radius, program the measuring cycle TCH PROBE 482. Select via input parameters by which of two methods the radius of a tool is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control measures the radius of each tooth with the aid of oriented spindle stops.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgTT** (No. 122700) Refer to your machine manual.

Cycle parameters

- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table
  0: The measured tool radius is written to column R of the tool table TOOL.T, and the tool compensation is set to DR=0. If there is already a value stored in TOOL.T, it will be overwritten.
  1: The measured tool radius is compared to the tool radius R from TOOL.T. It then calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T)
  2: The measured tool radius is compared to the tool radius R from TOOL.T. The TNC calculates the deviation from the stored value and enters it in Q parameter Q116. Nothing is entered under R or DR in the tool table.
- **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth)

NC blocks

6 TOOL CALL 12 Z
7 TCH PROBE 482 CAL. TOOL RADIUS
Q340=1 ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH

21.9 Measuring tool length and radius (Cycle 483, Option 17)

Cycle run

To measure both the length and radius of a tool, program the measuring cycle TCH PROBE 483. This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Via input parameters you can select the desired type of measurement:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for Cycles as well as 481 and 482.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgTT** (No. 122700) Refer to your machine manual.

Cycle parameters

- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table
  0: The measured tool length and measured tool radius are written to columns L and R of the tool table TOOL.T, and the tool compensation is set to DL=0 and DR=0. If there is already a value stored in TOOL.T, it will be overwritten.
  1: The measured tool length and measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The TNC calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in Q parameters Q115 and Q116. If the delta value is greater than the permissible tool length or radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T)
  2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The TNC calculates the deviation from the stored value and enters them in Q parameters Q115 and Q116. Nothing is entered under L, R, DL, or DR in the tool table.
- **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth)

NC blocks

6 TOOL CALL 12 Z	
7 TCH PROBE 483 MEASURE TOOL	
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH

22

Tables and Overviews

22.1 Machine-specific user parameters

Application

The parameter values are entered in the **configuration editor**.

> Refer to your machine manual.
>
> The machine tool builder can additionally make some machine-specific machine parameters available as user parameters, so that the user can configure the functions that are available.

The machine parameters are grouped as parameter objects in a tree structure in the configuration editor. Each parameter object has a name (e.g. **Settings for screen displays**) that gives information about the parameters it contains. A parameter object, also called "entity," is marked with an **E** in the folder symbol in the tree structure. Some machine parameters have a key name to identify them unambiguously. The key name assigns the parameter to a group (e.g. X for X axis). The respective group folder bears the key name and is marked by a **K** in the folder symbol.

> Operating notes:
>
> - The icons of not yet active parameters and objects appear dimmed. These can be activated with the **MORE FUNCTIONS** and **INSERT** soft key.
> - The control saves a modification list of the last 20 changes to the configuration data. To restore modifications, select the corresponding line and press the **MORE FUNCTIONS** and **CANCEL CHANGE** soft keys.

Changing the display of the parameters

If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts.

Proceed as follows in order to have the actual system names of the parameters be shown:

- Press the **Screen layout** key

DISPLAY THE SYS- TEM NAME

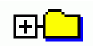
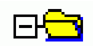
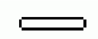
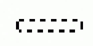
- Press the **SHOW SYSTEM NAME** soft key

Follow the same procedure to return to the standard display.

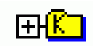
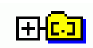
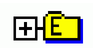
Calling the configuration editor and changing parameters

- Select the **Programming** operating mode
- Press the **MOD** key
- Enter the code number **123**
- Changing parameters
- Press the **END** soft key to exit the configuration editor
- Confirm changes with the **SAVE** soft key

The icon at the beginning of each line in the parameter tree shows additional information about this line. The icons have the following meanings:

-  Branch exists but is closed
-  Branch is open
-  Empty object, cannot be opened
-  Initialized machine parameter
-  Uninitialized (optional) machine parameter
-  Can be read but not edited
-  Can neither be read nor edited

The type of the configuration object is identified by its folder symbol:

-  Key (group name)
-  List
-  Entity (parameter object)

Displaying help texts

The **HELP** key enables you to call a help text for each parameter object or attribute.

If the help text does not fit on one page (1/2 is then displayed at the upper right, for example), press the **HELP PAGE** soft key to scroll to the second page.

As well as the Help text, other information is displayed, e.g. unit of measurement, initial value, selection list. If the selected machine parameter matches a parameter in the previous control model, the corresponding MP number is displayed.

Parameter list

Parameter settings

DisplaySettings

Settings for screen display

Sequence of displayed axes

[0] to [5]

Depends on available axes

Sequence of the displayed axes in the REF display

[0] to [5]

Depends on available axes

Type of position display in position window

NOMINAL

ACTUAL

REF ACTL

REF NOML

LAG

ACTDST

REFDST

M 118

Type of position display in status display

NOMINAL

ACTUAL

REF ACTL

REF NOML

LAG

ACTDST

REFDST

M 118

Definition of decimal separator for position display

. point

, comma

Display of feed rate in operating mode Manual operation

at axis key: Only display feed rate if axis direction key is pressed

always minimum: Always display feed rate

Display of spindle position in the position display

during closed loop: Only display spindle position if spindle is in position control

during closed loop and M5: Display spindle position if spindle is in position control and with M5

Show or hide soft key preset table

True: Soft key preset table is not displayed

Parameter settings
False: Display soft key preset table Font size with program display **FONT_APPLICATION_SMALL** **FONT_APPLICATION_MEDIUM** Sequence of icons in the display **[0] to [9]** **Depends on activated options**
DisplaySettings Display step for individual axes List of all available axes Display step for position display in mm or degrees **0.1** **0.05** **0.01** **0.005** **0.001** **0.0005** **0.0001** Display step for position display in inches **0.005** **0.001** **0.0005** **0.0001**
DisplaySettings Definition of unit of measure valid for the display **metric: Use metric system** **inch: Use inch system**
DisplaySettings Format of NC programs and display of cycles Program input in HEIDENHAIN Klartext conversational text or in DIN/ISO **HEIDENHAIN: Program input in operating mode MDI in Klartext conversational text dialog** **ISO: Program input in Positioning with MDI mode of operation in DIN/ISO**

Parameter settings

DisplaySettings

Setting the NC and PLC dialog language

NC dialog language

ENGLISH
GERMAN
CZECH
FRENCH
ITALIAN
SPANISH
PORTUGUESE
SWEDISH
DANISH
FINNISH
DUTCH
POLISH
HUNGARIAN
RUSSIAN
CHINESE
CHINESE_TRAD
SLOVENIAN
KOREAN
NORWEGIAN
ROMANIAN
SLOVAK
TURKISH

PLC dialog language

See NC dialog language

PLC error message language

See NC dialog language

Help language

See NC dialog language

Parameter settings

DisplaySettings
  Behavior with control start-up
    Acknowledge "Power interrupted" message
      TRUE: Control start-up is not continued until the message has been acknowledged
      FALSE: "Power interrupted" message not displayed

DisplaySettings
  Display mode for time display
    Selection for display mode in the time display
      Analog
      Digital
      Logo
      Analog and Logo
      Digital and Logo
      Analog on Logo
      Digital on Logo

DisplaySettings
  Link row On/Off
    Display setting for link row
      OFF: Deactivate the information line in the operating mode line
      ON: Activate the information line in the operating mode line

---

DisplaySettings
  Settings for 3-D display
    Model type of 3-D display
      3-D (compute-intensive): Model display for complex machining operations with undercuts
      2.5-D: Model display for 3-axis machining operations
      No Model: Model display is disabled

    Model quality of the 3-D display
      very high: High resolution; Block end points can be displayed
      high: High resolution
      medium: Medium resolution
      low: Low resolution

    Reset tool paths in new BLK form
      ON: With new BLK form in the test run, the tool paths are reset
      OFF: With new BLK form in the test run, the tool paths are not reset

---

Parameter settings

DisplaySettings

Settings for the position display

Position display

with TOOL CALL DL

As Tool Length: The programmed oversize DL is considered as the tool length modification for display of the workpiece-based position

As Workpiece Oversize: The programmed oversize DL is considered as the workpiece oversize for display of the workpiece-based position

---

DisplaySettings

Settings for the table editor

Behavior when deleting tools from the pocket table

DISABLED: Deletion of the tool is not possible

WITH_WARNING: Deletion of the tool is possible, must be confirmed

WITHOUT_WARNING: Deletion of the tool is possible without needing to be confirmation

Behavior when deleting index entries of a tool

ALWAYS_ALLOWED: Deletion of index entries is always possible

TOOL_RULES: The behavior depends on the setting of the parameter "Behavior when deleting tools from the pocket table"

Show the RÜCKS. SPALTE T soft key

TRUE: The soft key is shown and all tools can be deleted from the tool memory by the user

FALSE: The soft key is not shown

Parameter settings

ProbeSettings

Configuration of tool measurement

TT140_1

M function for spindle orientation

-1: Spindle orientation directly by NC

0: Function inactive

1 to 999: Number of M function for spindle orientation

Probing routine

MultiDirections: Probing from several directions

SingleDirection: Probing from one direction

Probing direction for tool radius measurement

X_Positive, Y_Positive, X_Negative, Y_Negative, Z_Positive, Z_Negative (depending on tool axis)

Distance between lower surface of tool and upper surface of stylus

0.001 to 99.9999 [mm]: Offset between stylus to tool

Rapid traverse in probing cycle

10 to 300 000 [mm/min]: Rapid traverse in probing cycle

Probing feed rate with tool measurement

1 to 3 000 [mm/min]: Probing feed rate with tool measurement

Calculation of probing feed rate

ConstantTolerance: Calculation of probing feed rate with constant tolerance

VariableTolerance: Calculation of probing feed rate with variable tolerance

ConstantFeed: Constant probing feed rate

Type of speed detection

Automatic: Determine speed automatically

MinSpindleSpeed: Use minimum spindle speed

Maximum permissible rotational speed the tool tip

1 to 129 [m/min]: Permissible rotational speed on cutter circumference

Maximum permissible speed with tool measurement

0 to 1 000 [rpm]: Maximum permissible speed

Maximum permissible measuring error with tool measurement

0.001 to 0.999 [mm]: First maximum permissible measuring error

Maximum permissible measuring error with tool measurement

0.001 to 0.999 [mm]: Second maximum permissible measuring error

NC stop during tool check

True: NC program is stopped if breakage tolerance is exceeded

Parameter settings

False: NC program is not stopped

NC stop during tool measurement

True: NC program is stopped if breakage tolerance is exceeded

False: NC program is not stopped

Modifying of tool table during tool check and measurement

AdaptOnMeasure: Table is modified after tool measurement

AdaptOnBoth: Table is modified after tool check and measurement

AdaptNever: Table is not modified after tool check and measurement

Configuration of a round stylus

TT140_1

Coordinates of the stylus center

[0]: X coordinate of stylus center referenced to machine datum

[1]: Y coordinate of stylus center referenced to machine datum

[2]: Z coordinate of stylus center referenced to machine datum

Safety clearance over stylus for pre-positioning

0.001 to 99 999.9999 [mm]: Safety clearance in tool axis direction

Safety zone around stylus for pre-positioning

0.001 to 99 999.9999 [mm]: Safety clearance in plane perpendicular to tool axis

Parameter settings

ChannelSettings

CH_NC

Active kinematics

Kinematics to be activated

List of machine kinematics

Kinematics to be activated with control start-up

List of machine kinematics

Determining the behavior of the NC program

Resetting the machining time with program start

True: Machining time is reset

False: Machining time is not reset

PLC signal for number of pending machining cycle

Dependent on machine manufacturer

Configuration of machining cycles

Overlap factor for pocket milling

0.001 to 1.414: Overlap factor for Cycle 4 POCKET MILLING

Behavior after machining a contour pocket

PosBeforeMachining: Position as before machining a cycle

ToolAxClearanceHeight: Position tool axis to clearance height

Display **Spindle ?** error message if M3/M4 is not active

on: Output error message

off: Do not output error message

Display **Enter negative depth** error message

on: Output error message

off: Do not output error message

M function for spindle orientation in machining cycles

-1: Spindle orientation directly via NC

0: Function inactive

1 to 999: Number of M function for spindle orientation

Do not display "**Plunging type not possible** error message

on: Error message is not displayed

off: Error message is displayed

Behavior of M7 and M8 with cycles 202 and 204

TRUE: At the end of cycle 202 and 204 the condition of M7 and M8 is restored before the cycle call

FALSE: At the end of cycle 202 and 204 the condition of M7 and M8 is not restored independently

Parameter settings

Do not show **Remaining material** warning

on: Warning is not displayed

off: Warning is displayed

CfgThreadSpindle

Potentiometer for feed rate during thread cutting

SpindlePotentiometer: During thread cutting, the potentiometer for shaft speed override is effective. The potentiometer for feed rate override is not active

FeedPotentiometer: During thread cutting, the potentiometer for feed rate override is effective. The potentiometer for shaft speed override is not active

Waiting time at reversal point in thread base

–999999999 to 999999999: The spindle stops for this time at the bottom of the thread before starting again in the opposite direction of rotation

Advanced switching time of spindle

–999999999 to 999999999: The spindle is stopped at this time before reaching the bottom of the thread

Limitation of spindle speed for Cycles 17, 207, and 18

TRUE: For small thread depths the spindles speed is limited to the extent that for about 1/3 of the time it runs at a constant speed

FALSE: No limitation of the spindle speed

Parameter settings

Settings for the NC editor

Creating backup files

TRUE: Create backup file after editing NC programs

FALSE: Create no backup file after editing NC programs

Cursor behavior after deleting lines

TRUE: Cursor is on previous line after deletion (iTNC behavior)

FALSE: Cursor is on subsequent line after deletion

Cursor behavior with the first and last line

TRUE: All-round cursors permitted at PGM beginning/end

FALSE: All-round cursors not permitted at PGM beginning/end

Line break with multi-line blocks

ALL: Always show lines completely

ACT: Only show lines of the active block completely

NO: Only show lines completely if the block is edited

Activate help graphics with cycle input

TRUE: Fundamentally always show help graphics during input

FALSE: Only show help graphics if the CYCLE HELP soft key is set to ON. The CYCLE HELP OFF/ON soft key is displayed in the Programming mode after pressing the "Screen layout" button

Behavior of soft key row following a cycle input

TRUE: Leave cycle soft key row active after a cycle definition

FALSE: Hide cycle soft key row after a cycle definition

Confirmation request before block is deleted

TRUE: Display confirmation request before deleting an NC block

FALSE: Do not display confirmation request before deleting an NC block

Line number up to which NC program is tested

100 to 50000: Program length for which the geometry is to be checked

ISO programming: Block number increment

0 to 250: Increment for generating ISO blocks in the program

Define programmable axes

TRUE: Use defined axis configuration

FALSE: Use default axis configuration XYZABCUVW

Line number up to which identical syntax elements are searched for

500 to 50000: Search for selected elements with up/down arrow keys

Behavior of PARAXMODE function with UVW axes

FALSE: PARAXMODE function permitted

TRUE: PARAXMODE function locked

Parameter settings
Settings for the file manager Display of dependent files **MANUAL: Dependent files are displayed** **AUTOMATIC: Dependent files are not displayed**
Path specifications for end users List with drives and/or directories **Drives and directories entered here are shown by the control in the file manager** FN 16 output path for execution **Path for FN 16 output if no path has been defined in the program** FN 16 output path for Programming and Test Run operating modes **Path for FN 16 output if no path has been defined in the program**
Serial Interface RS232 **Further information:** "Setting up data interfaces", page 479

22.2 Connector pin layout and connection cables for data interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices

> The interface complies with the requirements of EN 50 178 for **Low voltage electrical separation.**

When using the 25-pin adapter block:

Control		Conn. cable 365725-xx			Adapter block 310085-01		Conn. cable 274545-xx		
Male	**Assignment**	**Female**	**Color**	**Female**	**Male**	**Female**	**Male**	**Color**	**Female**
1	Do not assign	1		1	1	1	1	White/ Brown	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.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.

When using the 9-pin adapter block:

Control		Conn. cable 355484-xx			Adapter block 363987-02		Conn. cable 366964-xx		
Male	**Assign-ment**	**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	White/ Green	8	8	8	8	White/ Green	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.

Non-HEIDENHAIN devices

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

It depends on the unit and the type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363987-02		Conn. cable 366964-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	White/Green	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	External shield	Hsg.

Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	

22.3 Technical Information

Technical Information

Explanation of symbols

- ■ Default
- □ Axis option
- **1** Advanced Function Set 1

Specifications	
Components	■ Operating panel ■ TFT color flat-panel display with soft keys
Program memory	■ 2 GB
Input resolution and display step	■ As fine as 0.1 µm for linear axes ■ Up to 0.0001° for rotary axes
Input range	■ Maximum 999 999 999 mm or 999 999 999°
Block processing time	■ 6 ms
Axis feedback control	■ Position loop resolution: Signal period of the position encoder/1024 ■ Cycle time of position controller: 3 ms ■ Cycle time of speed controller: 200 µs
Range of traverse	■ Maximum 100 m (3937 inches)
Spindle speed	■ Maximum 100,000 rpm (analog speed command signal)
Error compensation	■ Linear and nonlinear axis error, backlash, thermal expansion ■ Static friction
Data interfaces	■ One each RS-232-C /V.24 max. 115 kilobaud ■ Expanded data interface with LSV-2 protocol for remote operation of the control through the data interface with the HEIDENHAIN software TNCremo ■ Ethernet interface 1000 BaseT ■ 3 x USB (1 x front USB 2.0; 2 x rear USB 3.0)
Ambient temperature	■ Operation: 5 °C to +45 °C ■ Storage: –35 °C to +65 °C

Input formats and units of control functions	
Positions, coordinates, chamfer lengths	–99 999.9999 to +99 999.9999 (5,4: number of digits before and after the decimal point) [mm]
Tool numbers	0 to 32 767.9 (5, 1)
Tool names	32 characters, enclosed by quotation marks with **TOOL CALL**. Permitted special characters: # $ % & . , - _
Detail 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/1]
Dwell time in Cycle 9	0 to 3600.000 (4, 3) [s]
Thread pitch in various cycles	-9.9999 to +9.9999 (2, 4) [mm]
Angle for spindle orientation	0 to 360.0000 (3, 4) [°]
Datum numbers in Cycle 7	0 to 2999 (4, 0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2, 6)
Miscellaneous functions M	0 to 999 (4, 0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	–99 999.9999 to +99 999.9999 (9, 6)
Labels (LBL) for program jumps	0 to 999 (5, 0)
Labels (LBL) for program jumps	Any text string in quotation marks (“ ”)
Number of program section repeats REP	1 to 65 534 (5, 0)
Error number in Q parameter function FN14	0 to 1199 (4, 0)

User functions

User functions	
Short description	■ Basic version: 3 axes plus closed-loop spindle □ 1. Additional axis for 4 axes plus closed-loop spindle □ 2. Additional axis for 5 axes plus closed-loop spindle
Program entry	In HEIDENHAIN conversational format
Position entry	■ Nominal positions for straight lines in Cartesian coordinates ■ Incremental or absolute dimensions ■ Display and entry in mm or inches
Tool tables	Multiple tool tables with any number of tools
Parallel operation	Creating a program with graphical support while another program is being run
Cutting data	Automatic calculation of spindle speed, cutting speed, feed per tooth and feed per revolution

User functions	
Program jumps	■ Subprograms ■ Program section repeats ■ Any desired program as subprogram
Machining cycles	■ Cycles for drilling, and conventional and rigid tapping ■ Roughing and finishing rectangular pockets ■ Cycles for pecking, reaming, boring, and counterboring ■ Roughing and finishing rectangular studs ■ Cycles for clearing level surfaces ■ Face milling ■ Cartesian and polar point patterns ■ OEM cycles (special cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	■ Datum shift, mirroring ■ Scaling factor (axis-specific)
Q parameters Programming with variables	■ Mathematical functions =, +, –, *, /, roots ■ Logical operations (=, ≠, <, >) ■ Calculating with parentheses ■ sin α, cos α, tan α , arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point ■ Functions for calculation of circles ■ String param.
Programming aids	■ Calculator ■ Complete list of all current error messages ■ Context-sensitive help function for error messages ■ TNCguide: The integrated help system. ■ Graphic support for the programming of cycles ■ Comment and structure blocks in the NC program
Teach-In	■ Actual positions can be transferred directly to the NC program
Test graphics Display modes	■ Graphical simulation before a program run, also while another program is being run ■ Plan view / projection in 3 planes / 3-D view ■ Detail enlargement

User functions	
Programming graphics	■ In **Programming** mode, the contour of the NC blocks is drawn on screen while they are being entered (2-D pencil-trace graphics), even if 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	■ Calculation of machining time in the **Test Run** operating mode ■ Display of the current machining time in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes
Preset management	■ For saving any presets
Contour, returning to	■ Block scan in any block in the program, returning the tool to the calculated nominal position to continue machining ■ Program interruption, contour departure and return
Datum tables	■ Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	■ Calibrating the touch probe ■ Presetting, manual ■ Tools can be measured automatically

Software options

Touch Probe Functions (option 17)	
Touch probe functions	**Touch probe cycles:** ■ Presetting in the **Manual operation** mode of operation ■ Tools can be measured automatically

HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component

Accessories

Accessories	
Electronic handwheels	■ HR 410: Portable handwheel ■ HR 130: Panel-mounted handwheel ■ HR 150: Up to three panel-mounted handwheels via handwheel adapter HRA 110
Touch probes	■ TS 248: 3-D touch trigger probe with cable connection ■ TS 260: 3-D touch trigger probe with cable connection ■ TT 160: 3-D touch trigger probe for tool measurement ■ KT 130: Simple touch trigger probe with cable connection

Fixed cycles

Cycle number	Cycle name	DEF active	CALL active
7	DATUM SHIFT	■	
8	MIRROR IMAGE	■	
9	DWELL TIME	■	
11	SCALING	■	
12	PGM CALL		■
13	ORIENTATION	■	
200	DRILLING		■
201	REAMING		■
202	BORING		■
203	UNIVERSAL DRILLING		■
204	BACK BORING		■
205	UNIVERSAL PECKING		■
206	TAPPING		■
207	RIGID TAPPING		■
220	POLAR PATTERN	■	
221	CARTESIAN PATTERN	■	
233	FACE MILLING		■
240	CENTERING		■
241	SINGLE-LIP D.H.DRLNG		■
247	PRESETTING	■	
251	RECTANGULAR POCKET		■
253	SLOT MILLING		■
256	RECTANGULAR STUD		■

Miscellaneous functions

M	Effect	Effective at block Start	End	Page
M0	Program STOP/Spindle STOP/Coolant OFF		■	338
M1	Optional program run STOP/Spindle STOP/Coolant OFF		■	464
M2	Stop program/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 1		■	338
M3	Spindle ON clockwise	■		338
M4	Spindle ON counterclockwise	■		
M5	Spindle STOP		■	
M6	Tool change/STOP program run (depending on machine parameter)/Spindle STOP		■	338
M8	Coolant ON	■		338
M9	Coolant OFF		■	
M13	Spindle ON clockwise/Coolant ON	■		338
M14	Spindle ON counterclockwise/Coolant on	■		
M30	Same function as M2		■	338
M89	Vacant miscellaneous function **or**	■		504
	cycle call, modally effective (depending on machine parameter)		■	
M91	Within the positioning block: Coordinates are referenced to machine datum	■		339
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position	■		339
M94	Reduce the rotary axis display to a value below 360°	■		341
M99	Blockwise cycle call		■	504
M136	Feed rate F in millimeters per spindle revolution	■		343
M137	Reset M136			
M140	Retraction from the contour in the tool-axis direction	■		344

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC 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
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.de

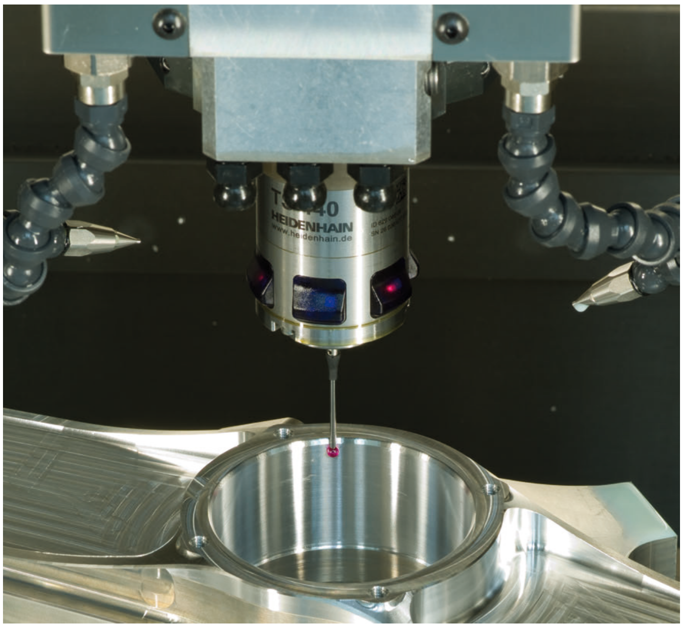
Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220	Signal transmission by cable
TS 440, TS 444	Infrared transmission
TS 640, TS 740	Infrared transmission

- Workpiece alignment
- Setting presets
- Workpiece measurement



Tool touch probes

TT 140	Signal transmission by cable
TT 449	Infrared transmission
TL	Non-contacting laser systems

- Tool measurement
- wear monitoring
- tool breakage detection

Original document
819494-24 · Ver04 · SW06 · 10/2017 · H · Printed in Germany