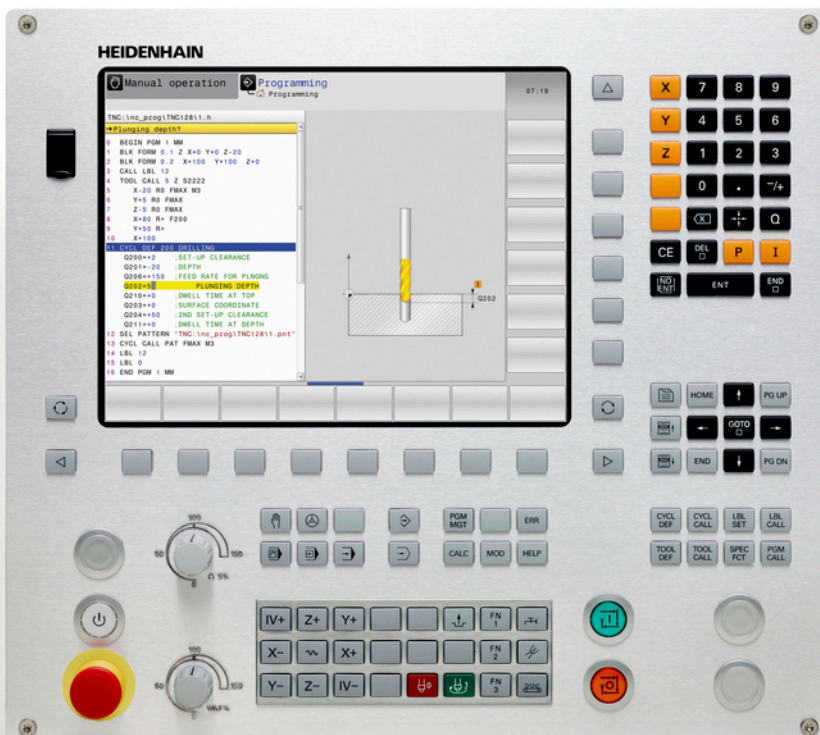




HEIDENHAIN



TNC 128

Conversational Programming
User's Manual





NC Software
771841-05

English (en)
9/2016






Controls of the TNC

Keys



Keys on visual display unit

Key	Function
	Selecting the screen layout
	Toggle the display between machining and programming modes
	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
	Select coordinate axes or enter them in a program
	Numbers
	Decimal point / Reverse algebraic sign
	Polar coordinate entry / Incremental values
	Q parameter programming / Q parameter status
	Capture actual position
	Skip dialog questions, delete words
	Confirm entry and resume dialog
	Conclude block and exit entry
	Clear entries or TNC error message
	Abort dialog, delete program section











Tool functions

Key	Function
	Define tool data in the program
	Call tool data





Manage programs and files, TNC functions

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



Navigation keys

Key	Function
 	Position the cursor
	Go directly to blocks, cycles and parameter functions
	Navigate to the program start or table start
	Navigate to the program end or end of a table line
	Navigate up one page
	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
 	Define and call cycles
 	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

The symbols used in this manual are described below.



This symbol indicates that important information about the function described must be considered.



This symbol indicates that there is one or more of the following risks when using the described function:

- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine



This symbol indicates a possibly dangerous situation that may cause injuries if not avoided.



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.



This symbol indicates that you can find detailed information about a function in another manual.

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

TNC model, software and features

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

TNC model	NC software number
TNC 128	771841-05

The machine manufacturer adapts the usable features of the TNC 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 TNC on your machine.

TNC 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, including HEIDENHAIN, offer programming courses for the TNCs. Participation in one of these courses is recommended to familiarize yourself thoroughly with the TNC 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: <ul style="list-style-type: none">■ Set the datum 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 TNC software are managed via the **Feature Content Level** upgrade functions. If you install a software update on your TNC 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 TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

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

- ▶ Operating mode **Programming**
- ▶ MOD function
- ▶ **LICENSE INFO** soft key

New functions

New functions 77184x-02

- New special operating mode RETRACT, see "Retraction after a power interruption", page 410
- New graphic simulation, "Graphics "
- New MOD function "tool usage file" within the machine settings group, see "Tool usage file", page 426
- New MOD function "set system time" within the systems settings group, see "Set the system time", page 427
- New MOD group "graphic settings", see "Graphic settings", page 422
- With the new cutting data calculator you can calculate the spindle speed and the feed rate, see "Cutting data calculator", page 160
- With the jump commands new if/then decisions have been introduced, see "Programming if-then decisions", page 241
- The character set of machining Cycle 225 Engraving has been expanded to include more characters and the diameter sign, see Cycle Programming User's Manual
- New machining Cycle 275 Trochoidal Milling, see Cycle Programming User's Manual
- New machining Cycle 233 Face Milling, see Cycle Programming User's Manual
- In drilling Cycles 200, 203 and 205, the parameter Q395 DEPTH REFERENCE has been introduced in order to evaluate the T ANGLE, see Cycle Programming User's Manual
- Probing Cycle 4 MEASURING IN 3-D has been introduced, see Cycle Programming User's Manual

Modified functions 77184x-02

- Up to 4 M functions are now allowed in an NC block, see "Fundamentals", page 296
- New soft keys for transferring values have been introduced in the pocket calculator, see "Operation", page 157
- The distance-to-go display can now also be displayed in the input system, see "Select the position display", page 428
- Several input parameters have been added to Cycle 241 SINGLE-LIP DEEP HOLE DRILLING, see Cycle Programming User's Manual
- Parameter Q305 NUMBER IN TABLE has been added to Cycle 404, see Cycle Programming User's Manual
- In the thread milling Cycles 26x, an approaching feed rate has been introduced, see Cycle Programming User's Manual
- In Cycle 205 Universal Deep Hole Drilling you can now use parameter Q208 to define a feed rate for retraction, see Cycle Programming User's Manual

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 219
- New **FEED DWELL** function for programming repeating dwell times, see "Dwell time FUNCTION FEED", page 317
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", page 254
- 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 580
- Protective zones can be defined in the MOD menu, see "Entering traverse limits", page 425
- Write protection is possible for single lines in the preset table, see "Saving the datums in the preset table", page 352
- New manual probing function for aligning a plane, see "Measuring 3-D basic rotation"
- CAD files can be opened without option number 42, see "CAD viewer", page 209

Modified functions 77184x-03

- FZ and FU feed rate input possible in the Tool Call block, see "Call tool data", page 193
- The input range of the DOC column in the pocket table has been expanded to 32 characters, see "Pocket table for tool changer", page 190
- 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 table is write-protected in Programming operating mode, see "Saving the datums in the preset table", page 352
- 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 92
- Manual calibration of the touch probe with fewer pre-positioning movements, see "Calibrating 3-D touch probes (option 17)", page 369
- 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 181
- In single block mode the control executes each point individually with point pattern cycles and CYCL CALL PAT, see "Program run", page 405
- Rebooting the control is no longer possible with the **END** key but with the **RESTART** soft key, see "Switch-off", page 335
- The control displays the contouring feed rate in manual mode, see "Spindle speed S, feed rate F and miscellaneous function M", page 349
- 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 580

New and modified cycle functions 77184x-03

- Cycle **253 SLOT MILLING** has been added (option 19)

New functions 77184x-05

- New function **FUNCTION DWELL** for programming a dwell time, see "Dwell time FUNCTION DWELL", page 331
- New function **FUNCTION S-PULSE** for programming pulsing shaft speeds, see "Pulsing spindle speed FUNCTION S-PULSE", page 315
- The column **KINEMATIC** has been added to the tool table, see "Enter tool data into the table", page 182
- The column **OVRTIME** has been added to the tool table, see "Enter tool data into the table", page 182
- 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 "Import and export tool data"
- Manual probe functions create a line in the preset table that does not yet exist, see "Writing measured values from the touch probe cycles to the preset table", page 368
- Manual probe functions can write in a password-protected line, see "Recording measured values from the touch probe cycles", page 366
- During a manual touch probe cycle, control can be transferred to the handwheel, see "Traverse movements with a handwheel with display", page 362
- 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 "Interrupt, stop or abort machining", page 406
- With active structuring the structure block can be edited in the structure window, see "Definition and applications", page 155
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", page 254
- The FN16 functions have been expanded, see "FN16: F-PRINT – Formatted output of text and Q parameter values", page 249
- The file saved with **SAVE AS** is now also found in the file management under **LAST FILES**, see "Editing a program", page 119
- If you save files with **SAVE AS**, you can select the target directory with the **SWITCH** soft key, see "Editing a 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 130
- File management displays vertical scrollbars and supports scrolling with the mouse, see "Calling the file manager", page 129

TNC model, software and features

- New machine parameter for recreating **M7** and **M8**, see "Machine-specific user parameters", page 580
- The function **STRLEN** checks whether a string parameter has been defined, see "Finding the length of a string parameter", page 286
- The function **SYSSTR** enables the NC software version to be read out, see "Reading system data", page 283
- 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 241
- Cylindrical workpiece blanks can now also be defined with a diameter instead of a radius, see "Define 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 155
- In operating modes **Program run full sequence**, **Program run single block** and **Positioning with manl.data input**, the font size can be set to the same size as the **Programming** operating mode, see "Machine-specific user parameters", page 580
- The functions in the **Positioning with manl.data input** mode were expanded and adapted for improved operation, see "Positioning with Manual Data Input", page 383
- 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 410
- In **Test run** operating mode a tool usage file can also be created without simulation, see "Tool usage test", page 196
- In the **Test run** mode of operation you can hide the rapid traverse movements by using the **FMAX PATHS** soft key, see "3-D view in the Test Run operating mode", page 395

- In the **Test run** operating mode you can reset the volume with the **RESET THE VOLUME MODEL** soft key, see "3-D view in the Test Run operating mode", page 395
- In the **Test run** operating mode you can reset the tool paths by using the **RESET TOOL PATHS** soft key, see "3-D view in the Test Run operating mode", page 395
- In **Test run** operating mode the soft key **MEASURING** displays the coordinates if you position the mouse on the graphics, see "3-D view in the Test Run operating mode", page 395
- In **Test run** operating mode the soft key **STOP AT** simulates up to a predefined block, see "Test run up to a certain block ", page 404
- Active basic transformation is shown in the status display on the **POS** tab, see "Positions and coordinates (POS tab)", page 90
- The status display now also shows the path of the active main program, see "Overview", page 88, see "General program information (PGM tab)", page 88
- Mid-program startup can now be continued, see "Any entry into program: Mid-program startup", page 412
- 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 339

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 185
- When importing tool tables, nonexistent tool types are imported as type undefined, see "Importing tool tables", page 188
- Tool names can now also include the special characters % and ,, see "Tool number, tool name", page 180
- When importing tool tables the numerical values are adopted from the **R-OFFS** column, see "Importing tool tables", page 188
- In the **LIFTOFF** column of the tool table the default is now **N**, see "Enter tool data into the table", page 182
- The **L** and **R** columns of the tool table are empty when a new tool is created, see "Editing the tool table", page 185
- In the tool table for the **RT** and **KINEMATIC** columns, the **SELECT** soft key is now available, see "Enter tool data into the table", page 182
- You cannot delete the tool data of tools still stored in the pocket table, see "Editing the tool table", page 185
- 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 363
- 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 349
- 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 350
- Soft key allocations were adapted for incremental positioning
- The values entered for the traverse limits are checked for validity, see "Entering traverse limits", page 425

- When the preset table 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 178
- 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 155
- 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 vacant again, see "Configuring the TNC", page 437
- 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 TNC", page 148
- 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 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 162
- In **Test run** and **Programming** operating modes the tool data is reset if a program is reselected or restarted with the **RESET + START** soft key
- In **Test run** operating mode the control displays the datum of the machine tool table as the reference point with **BLANK IN WORK SPACE**, see "Showing the workpiece blank in the working space", page 399
- After modification of the active datum, resuming the program is only possible after **GOTO** or mid-program startup, see "Moving the machine axes during an interruption", page 408
- Mid-program startup operation and dialog guidance has been improved, also for pallet tables, see "Any entry into program: Mid-program startup", page 412

New and modified cycle functions 77184x-05

- For cycle 247: SET DATUM, the datum number from the preset table can be selected with the corresponding parameter
- With cycle 200 and 203 the behavior of the dwell time at top has been adapted
- Cycle 205 performs deburring on the coordinate surface
- In cycles 481 to 483 the parameter Q340 was expanded with the input option "2". This enables a tool control without making any change in the tool table

Contents

1	First Steps with the TNC 128.....	59
2	Introduction.....	79
3	Fundamentals, file management.....	107
4	Programming aids.....	151
5	Tools.....	177
6	Programming tool movements.....	201
7	Data transfer from CAD files.....	207
8	Subprograms and program section repeats.....	211
9	Programming Q parameters.....	229
10	Miscellaneous functions.....	295
11	Special functions.....	305
12	Manual Operation and Setup.....	333
13	Positioning with Manual Data Input.....	383
14	Test Run and Program Run.....	389
15	MOD Functions.....	419
16	Fundamentals / Overviews.....	451
17	Cycles: Drilling cycles / thread cycles.....	475
18	Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling.....	511
19	Cycles: Coordinate Transformations.....	535
20	Cycles: Special Functions.....	551
21	Touch probe cycles.....	557
22	Tables and Overviews.....	579

1	First Steps with the TNC 128.....	59
1.1	Overview.....	60
1.2	Machine switch-on.....	60
	Acknowledging the power interruption and moving to the reference points.....	60
1.3	Programming the first part.....	61
	Selecting the correct operating mode.....	61
	The most important TNC keys.....	61
	Opening a new program/file management.....	62
	Defining a workpiece blank.....	63
	Program layout.....	64
	Programming a simple contour.....	65
	Creating a cycle program.....	67
1.4	Graphically testing the first part.....	70
	Selecting the correct operating mode.....	70
	Selecting the tool table for the test run.....	70
	Choosing the program you want to test.....	71
	Selecting the screen layout and the view.....	71
	Starting the test run.....	72
1.5	Setting up tools.....	73
	Selecting the correct operating mode.....	73
	Preparing and measuring tools.....	73
	The tool table TOOL.T.....	74
	The pocket table TOOL_PTCH.....	75
1.6	Workpiece setup.....	76
	Selecting the correct operating mode.....	76
	Clamping the workpiece.....	76
	Datum setting with a 3-D touch probe (option number 17).....	77
1.7	Running the first program.....	78
	Selecting the correct operating mode.....	78
	Choosing the program you want to run.....	78
	Start the program.....	78

2	Introduction.....	79
2.1	The TNC 128.....	80
	HEIDENHAIN Klartext.....	80
	Compatibility.....	80
2.2	Visual display unit and operating panel.....	81
	Display screen.....	81
	Set screen layout.....	81
	Control panel.....	82
2.3	Modes of operation.....	83
	Manual Operation and El. Handwheel.....	83
	Positioning with Manual Data Input.....	83
	Programming.....	83
	Test Run.....	84
	Program Run, Full Sequence and Program Run, Single Block.....	84
2.4	Status displays.....	85
	General status display.....	85
	Additional status displays.....	87
2.5	Window manager.....	93
	Overview of the task bar.....	94
	Portscan.....	96
	Remote Service.....	97
	SELinux security software.....	99
	VNC.....	100
	Backup and restore.....	102
2.6	Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels.....	104
	3-D touch probes.....	104
	HR electronic handwheels.....	105

3	Fundamentals, file management.....	107
3.1	Fundamentals.....	108
	Position encoders and reference marks.....	108
	Reference system.....	108
	Reference system of milling machines.....	109
	Designation of the axes on milling machines.....	109
	Absolute and incremental workpiece positions.....	110
	Selecting the datum.....	111
3.2	Opening programs and entering.....	112
	Structure of an NC program in HEIDENHAIN Klartext.....	112
	Define the blank: BLK FORM.....	113
	Opening a new part program.....	115
	Programming tool movements in Klartext.....	116
	Actual position capture.....	118
	Editing a program.....	119
	The TNC search function.....	123
3.3	File management: Basics.....	124
	Files.....	124
	Displaying externally generated files on the TNC.....	126
	Data Backup.....	126

3.4 Working with the file manager.....	127
Directories.....	127
Paths.....	127
Overview: Functions of the file manager.....	128
Calling the file manager.....	129
Selecting drives, directories and files.....	130
Creating a new directory.....	132
Create new file.....	132
Copying a single file.....	132
Copying files into another directory.....	133
Copying a table.....	134
Copying a directory.....	135
Choose one of the last files selected.....	135
Deleting a file.....	136
Deleting a directory.....	136
Tag files.....	137
Renaming a file.....	137
Sort files.....	138
Additional functions.....	138
Additional tools for management of external file types.....	139
Data transfer to or from an external data carrier.....	146
The TNC in a network.....	147
USB devices on the TNC.....	148

4	Programming aids.....	151
4.1	Screen keyboard.....	152
	Entering text with the screen keyboard.....	152
4.2	Adding comments.....	153
	Application.....	153
	Add comments.....	153
	Functions for editing of the comment.....	153
4.3	Display of NC programs.....	154
	Syntax highlighting.....	154
	Scrollbar.....	154
4.4	Structuring programs.....	155
	Definition and applications.....	155
	Displaying the program structure window / Changing the active window.....	155
	Inserting a structure block in the program window.....	156
	Selecting blocks in the program structure window.....	156
4.5	Calculator.....	157
	Operation.....	157
4.6	Cutting data calculator.....	160
	Application.....	160
4.7	Programming graphics.....	162
	Generate/do not generate graphics during programming.....	162
	Generating a graphic for an existing program.....	163
	Block number display ON/OFF.....	164
	Erasing the graphic.....	164
	Showing grid lines.....	164
	Magnification or reduction of details.....	165

4.8 Error messages..... 166

Display of errors.....	166
Open the error window.....	166
Closing the error window.....	166
Detailed error messages.....	167
Soft key: INTERNAL INFO.....	167
Soft key FILTER.....	167
Clearing errors.....	168
Error log.....	168
Keystroke log.....	169
Informational texts.....	170
Save service files.....	170
Calling the TNCguide help system.....	170

4.9 TNCguide context-sensitive help system..... 171

Application.....	171
Working with TNCguide.....	172
Downloading current help files.....	175

5	Tools.....	177
5.1	Entering tool-related data.....	178
	Feed rate F.....	178
	Spindle speed S.....	179
5.2	Tool data.....	180
	Requirements for tool compensation.....	180
	Tool number, tool name.....	180
	Tool length L.....	180
	Tool radius R.....	180
	Delta values for lengths and radii.....	181
	Entering tool data into the program.....	181
	Enter tool data into the table.....	182
	Importing tool tables.....	188
	Overwriting tool data from an external PC.....	189
	Pocket table for tool changer.....	190
	Call tool data.....	193
	Tool change.....	195
	Tool usage test.....	196
5.3	Tool compensation.....	198
	Introduction.....	198
	Tool length compensation.....	198
	Tool radius compensation with paraxial positioning blocks.....	199

6 Programming tool movements..... 201

6.1 Fundamentals..... 202

 Tool movements in the program.....202

 Miscellaneous functions M.....203

 Subprograms and program section repeats..... 203

 Programming with Q parameters..... 203

6.2 Tool movements..... 204

 Programming tool movements for workpiece machining..... 204

 Capture actual position..... 205

 Example: Linear movement..... 206

7	Data transfer from CAD files.....	207
7.1	CAD viewer and.....	208
	Fundamentals of the CAD viewer and.....	208
7.2	CAD viewer.....	209
	Application.....	209

8	Subprograms and program section repeats.....	211
8.1	Labeling subprograms and program section repeats.....	212
	Label.....	212
8.2	Subprograms.....	213
	Operating sequence.....	213
	Programming notes.....	213
	Program the subprogram.....	213
	Calling a subprogram.....	214
8.3	Program-section repeats.....	215
	Label.....	215
	Operating sequence.....	215
	Programming notes.....	215
	Programming a program section repeat.....	216
	Calling a program section repeat.....	216
8.4	Any desired program as subprogram.....	217
	Overview of the soft keys.....	217
	Operating sequence.....	218
	Programming notes.....	218
	Calling any program as a subprogram.....	219
8.5	Nesting.....	221
	Types of nesting.....	221
	Nesting depth.....	221
	Subprogram within a subprogram.....	222
	Repeating program section repeats.....	223
	Repeating a subprogram.....	224
8.6	Programming examples.....	225
	Example: Groups of holes.....	225
	Example: Group of holes with several tools.....	227

9	Programming Q parameters.....	229
9.1	Principle and overview of functions.....	230
	Programming notes.....	232
	Calling Q parameter functions.....	233
9.2	Part families—Q parameters in place of numerical values.....	234
	Application.....	234
9.3	Describing contours with mathematical functions.....	235
	Application.....	235
	Overview.....	235
	Programming fundamental operations.....	236
9.4	Angle functions.....	238
	Definitions.....	238
	Programming trigonometric functions.....	238
9.5	Calculation of circles.....	239
	Application.....	239
9.6	If-then decisions with Q parameters.....	240
	Application.....	240
	Unconditional jumps.....	240
	Abbreviations used:.....	240
	Programming if-then decisions.....	241
9.7	Checking and changing Q parameters.....	242
	Procedure.....	242
9.8	Additional functions.....	244
	Overview.....	244
	FN 14: ERROR: Displaying error messages.....	245
	FN16: F-PRINT – Formatted output of text and Q parameter values.....	249
	FN 18: SYSREAD – Reading system data.....	254
	FN 19: PLC – Transfer values to the PLC.....	262
	FN 20: WAIT FOR – NC and PLC synchronization.....	262
	FN 29: PLC – Transfer values to the PLC.....	263
	FN 37: EXPORT.....	263
	FN 38: SEND – Send information from NC program.....	263

9.9 Accessing tables with SQL commands..... 264

Introduction.....	264
A transaction.....	265
Programming SQL commands.....	267
Overview of the soft keys.....	268
SQL BIND.....	269
SQL SELECT.....	270
SQL FETCH.....	272
SQL UPDATE.....	273
SQL INSERT.....	273
SQL COMMIT.....	274
SQL ROLLBACK.....	274

9.10 Entering formulas directly..... 275

Entering formulas.....	275
Rules for formulas.....	277
Example of entry.....	278

9.11 String parameters..... 279

String processing functions.....	279
Assign string parameters.....	280
Chain-linking string parameters.....	280
Converting a numerical value to a string parameter.....	281
Copying a substring from a string parameter.....	282
Reading system data.....	283
Converting a string parameter to a numerical value.....	284
Testing a string parameter.....	285
Finding the length of a string parameter.....	286
Compare alphabetic priority.....	287
Reading out machine parameters.....	288

9.12 Preassigned Q parameters..... 291

Values from the PLC: Q100 to Q107.....	291
Active tool radius: Q108.....	291
Tool axis: Q109.....	291
Spindle status: Q110.....	292
Coolant on/off: Q111.....	292
Overlap factor: Q112.....	292
Unit of measurement for dimensions in the program: Q113.....	292
Tool length: Q114.....	292
Coordinates after probing during program run.....	293
Deviation between actual value and nominal value during automatic tool measurement with the TT 130.....	293

10	Miscellaneous functions.....	295
10.1	Enter miscellaneous functions M.....	296
	Fundamentals.....	296
10.2	Miscellaneous functions for program run inspection, spindle and coolant.....	298
	Overview.....	298
10.3	Miscellaneous functions for coordinate entries.....	299
	Programming machine-referenced coordinates: M91/M92.....	299
	Reducing display of a rotary axis to a value less than 360°: M94.....	301
10.4	Miscellaneous functions for path behavior.....	302
	Feed rate factor for plunging movements: M103.....	302
	Feed rate in millimeters per spindle revolution: M136.....	303
	Retraction from the contour in the tool-axis direction: M140.....	304

11 Special functions.....	305
11.1 Overview of special functions.....	306
Main menu for SPEC FCT special functions.....	306
Program defaults menu.....	307
Functions for contour and point machining menu.....	307
Menu of various conversational functions.....	308
11.2 Freely definable tables.....	309
Fundamentals.....	309
Creating a freely definable table.....	309
Editing the table format.....	310
Switching between table and form view.....	311
FN 26: TABOPEN – Open a freely definable table.....	312
FN 27: TABWRITE – Write to a freely definable table.....	313
FN 28: TABREAD – Read from a freely definable table.....	314
Customize table view.....	314
11.3 Pulsing spindle speed FUNCTION S-PULSE.....	315
Program pulsing spindle speed.....	315
Reset pulsing spindle speed.....	316
11.4 Dwell time FUNCTION FEED.....	317
Programming dwell time.....	317
Resetting dwell time.....	318
11.5 File functions.....	319
Application.....	319
Defining file functions.....	319
11.6 Definition of a datum shift.....	320
Overview.....	320
TRANS DATUM AXIS.....	320
TRANS DATUM TABLE.....	321
TRANS DATUM RESET.....	322

11.7 Creating text files.....323

Application.....	323
Opening and exiting a text file.....	323
Editing texts.....	324
Deleting and re-inserting characters, words and lines.....	324
Editing text blocks.....	325
Finding text sections.....	326

11.8 Tool carrier management..... 327

Fundamentals.....	327
Save tool carrier templates.....	327
Assign input parameters to tool carriers.....	328
Allocate parameterized tool carriers.....	330

11.9 Dwell time FUNCTION DWELL..... 331

Programming dwell time.....	331
-----------------------------	-----

12 Manual Operation and Setup.....	333
12.1 Switch-on, switch-off.....	334
Switch-on.....	334
Switch-off.....	335
12.2 Moving the machine axes.....	336
Note.....	336
Moving the axis with the axis direction keys.....	336
Incremental jog positioning.....	337
Traverse with the HR 410 electronic handwheel.....	338
Moving with the electronic display handwheels.....	339
12.3 Spindle speed S, feed rate F and miscellaneous function M.....	349
Application.....	349
Entering values.....	349
Adjusting spindle speed and feed rate.....	350
Feed rate limit F MAX.....	350
12.4 Datum management with the preset table.....	351
Note.....	351
Saving the datums in the preset table.....	352
Activating the datum.....	358
12.5 Datum setting without a 3-D touch probe.....	359
Note.....	359
Preparation.....	359
Datum setting with an end mill.....	359
Using touch probe functions with mechanical probes or measuring dials.....	360
12.6 Using a 3-D touch probe (option 17).....	361
Overview.....	361
Functions in touch probe cycles.....	363
Selecting the probing cycle.....	365
Recording measured values from the touch probe cycles.....	366
Writing measured values from the touch probe cycles to a datum table.....	367
Writing measured values from the touch probe cycles to the preset table.....	368

12.7 Calibrating 3-D touch probes (option 17)..... 369

Introduction..... 369

Calibrating the effective length..... 370

Calibrating the effective radius and compensating center misalignment..... 371

Displaying calibration values..... 374

12.8 Datum setting with a 3-D touch probe (option number 17)..... 375

Overview..... 375

Datum setting on any axis..... 375

Circle center as datum..... 376

Setting a center line as datum..... 379

Measuring workpieces with a 3-D touch probe..... 380

13 Positioning with Manual Data Input.....	383
13.1 Programming and executing simple machining operations.....	384
Positioning with manual data input (MDI).....	385
Protecting programs in \$MDI.....	388

14 Test Run and Program Run.....	389
14.1 Graphics.....	390
Application.....	390
Speed of the setting test runs.....	391
Overview: Display modes.....	392
Plan view.....	392
Projection in three planes.....	393
3-D view.....	394
Repeating graphic simulation.....	397
Tool display.....	397
Measurement of machining time.....	398
14.2 Showing the workpiece blank in the working space.....	399
Application.....	399
14.3 Functions for program display.....	400
Overview.....	400
14.4 Test run.....	401
Application.....	401
Execute test run.....	403
Test run up to a certain block.....	404
14.5 Program run.....	405
Application.....	405
Running a part program.....	405
Interrupt, stop or abort machining.....	406
Moving the machine axes during an interruption.....	408
Resuming program run after an interruption.....	409
Retraction after a power interruption.....	410
Any entry into program: Mid-program startup.....	412
Returning to the contour.....	416
14.6 Optional block skip.....	417
Application.....	417
Inserting the "/" character.....	417
Erasing the "/" character.....	417

14.7 Optional program-run interruption.....418

Application.....418

15 MOD Functions.....	419
15.1 MOD function.....	420
Selecting MOD functions.....	420
Changing the settings.....	420
Exiting MOD functions.....	420
Overview of MOD functions.....	421
15.2 Graphic settings.....	422
15.3 Machine settings.....	423
External access.....	423
Entering traverse limits.....	425
Tool usage file.....	426
Select kinematics.....	426
15.4 System settings.....	427
Set the system time.....	427
15.5 Select the position display.....	428
Application.....	428
15.6 Setting the unit of measure.....	429
Application.....	429
15.7 Displaying operating times.....	429
Application.....	429
15.8 Software numbers.....	430
Application.....	430
15.9 Enter the code number.....	430
Application.....	430

15.10 Setting up data interfaces.....431

Serial interfaces on the TNC 128.....	431
Application.....	431
Setting the RS-232 interface.....	431
Set BAUD RATE (baud rate no. 106701).....	431
Set protocol (protocol no. 106702).....	432
Set data bits (dataBits no. 106703).....	432
Check parity (parity no. 106704).....	432
Set stop bits (stopBits no. 106705).....	432
Set handshake (flowControl no. 106706).....	433
File system for file operation (fileSystem no. 106707).....	433
Block check character (bccAvoidCtrlChar no. 106708).....	433
Condition of RTS line (rtsLow no. 106709).....	433
Define behavior after receipt of ETX (noEotAfterEtx no. 106710).....	434
Settings for the transmission of data using PC software TNCserver.....	434
Setting the operating mode of the external device (fileSystem).....	435
Software for data transfer.....	435

15.11 Ethernet interface.....437

Introduction.....	437
Connection possibility.....	437
Configuring the TNC.....	437

15.12 Firewall.....443

Application.....	443
------------------	-----

15.13 Configuring the HR 550FS wireless handwheel.....446

Application.....	446
Assigning the handwheel to a specific handwheel holder.....	446
Setting the transmission channel.....	447
Selecting the transmitter power.....	447
Statistical data.....	448

15.14 Load machine configuration.....449

Application.....	449
------------------	-----

16 Fundamentals / Overviews.....	451
16.1 Introduction.....	452
16.2 Available Cycle Groups.....	453
Overview of fixed cycles.....	453
16.3 Working with fixed cycles.....	454
Machine-specific cycles.....	454
Defining a cycle using soft keys.....	455
Defining a cycle using the GOTO function.....	455
Calling a cycle.....	456
16.4 Program defaults for cycles.....	458
Overview.....	458
Entering GLOBAL DEF.....	458
Using GLOBAL DEF information.....	459
Global data valid everywhere.....	459
Global data for drilling operations.....	460
Global data for milling operations with pocket cycles 25x.....	460
Global data for milling operations with contour cycles.....	460
Global data for positioning behavior.....	461
Global data for probing functions.....	461
16.5 PATTERN DEF pattern definition.....	462
Application.....	462
Entering PATTERN DEF.....	462
Using PATTERN DEF.....	463
Defining individual machining positions.....	463
Defining a single row.....	464
Defining a single pattern.....	465
Defining individual frames.....	466
Defining a full circle.....	467
Defining a pitch circle.....	467
16.6 POLAR PATTERN (Cycle 220).....	468
Cycle run.....	468
Please note while programming:.....	468
Cycle parameters.....	469

16.7 LINEAR PATTERN (Cycle 221)..... 470

Cycle run..... 470

Please note while programming:.....470

Cycle parameters..... 471

16.8 Point tables.....472

Application.....472

Creating a point table..... 472

Hiding single points from the machining process..... 473

Selecting a point table in the program.....473

Calling a cycle in connection with point tables..... 474

17 Cycles: Drilling cycles / thread cycles.....	475
17.1 Fundamentals.....	476
Overview.....	476
17.2 CENTERING (Cycle 240).....	477
Cycle run.....	477
Please note while programming:.....	477
Cycle parameters.....	478
17.3 DRILLING (Cycle 200).....	479
Cycle run.....	479
Please note while programming:.....	479
Cycle parameters.....	480
17.4 REAMING (Cycle 201).....	481
Cycle run.....	481
Please note while programming:.....	481
Cycle parameters.....	482
17.5 BORING (Cycle 202).....	483
Cycle run.....	483
Please note while programming:.....	484
Cycle parameters.....	485
17.6 UNIVERSAL DRILLING (Cycle 203).....	486
Cycle run.....	486
Please note while programming:.....	486
Cycle parameters.....	487
17.7 BACK BORING (Cycle 204).....	489
Cycle run.....	489
Please note while programming:.....	490
Cycle parameters.....	491
17.8 UNIVERSAL PECKING (Cycle 205).....	493
Cycle run.....	493
Please note while programming:.....	494
Cycle parameters.....	495

17.9 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)..... 497

Cycle run.....	497
Please note while programming:.....	497
Cycle parameters.....	498

17.10 Programming Examples..... 500

Example: Drilling cycles.....	500
Example: Using drilling cycles in connection with PATTERN DEF.....	501

17.11 TAPPING with a floating tap holder (Cycle 206)..... 503

Cycle run.....	503
Please note while programming:.....	503
Cycle parameters.....	504

17.12 RIGID TAPPING without a floating tap holder (Cycle 207)..... 505

Cycle run.....	505
Please note while programming:.....	506
Cycle parameters.....	507
Retracting after a program interruption.....	507

17.13 Programming Examples..... 508

Example: Thread milling.....	508
------------------------------	-----

18 Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling.....	511
18.1 Fundamentals.....	512
Overview.....	512
18.2 RECTANGULAR POCKET (Cycle 251).....	513
Cycle run.....	513
Please note while programming:.....	514
Cycle parameters.....	515
18.3 SLOT MILLING (Cycle 253, DIN/ISO: G253).....	517
Cycle run.....	517
Please note while programming:.....	518
Cycle parameters.....	519
18.4 RECTANGULAR STUD (Cycle 256).....	521
Cycle run.....	521
Please note while programming:.....	522
Cycle parameters.....	523
18.5 FACE MILLING (Cycle 233).....	525
Cycle run.....	525
Please note while programming:.....	529
Cycle parameters.....	530
18.6 Programming Examples.....	533
Example: Milling pockets, studs.....	533

19 Cycles: Coordinate Transformations.....	535
19.1 Fundamentals.....	536
Overview.....	536
Effectiveness of coordinate transformations.....	536
19.2 DATUM SHIFT (Cycle 7).....	537
Effect.....	537
Cycle parameters.....	537
19.3 DATUM SHIFT with datum tables (Cycle 7).....	538
Effect.....	538
Please note while programming:.....	539
Cycle parameters.....	539
Selecting a datum table in the part program.....	540
Editing the datum table in the Programming mode of operation.....	540
Configuring a datum table.....	542
Leaving a datum table.....	542
Status displays.....	542
19.4 DATUM SETTING (Cycle 247).....	543
Effect.....	543
Please note before programming:.....	543
Cycle parameters.....	543
19.5 MIRRORING (Cycle 8).....	544
Effect.....	544
Cycle parameters.....	544
19.6 SCALING (Cycle 11).....	545
Effect.....	545
Cycle parameters.....	545
19.7 AXIS-SPECIFIC SCALING (Cycle 26).....	546
Effect.....	546
Please note while programming:.....	546
Cycle parameters.....	547
19.8 Programming Examples.....	548
Example: Groups of holes.....	548

20 Cycles: Special Functions.....	551
20.1 Fundamentals.....	552
Overview.....	552
20.2 DWELL TIME (Cycle 9).....	553
Function.....	553
Cycle parameters.....	553
20.3 PROGRAM CALL (Cycle 12).....	554
Cycle function.....	554
Please note while programming:.....	554
Cycle parameters.....	555
20.4 SPINDLE ORIENTATION (Cycle 13).....	556
Cycle function.....	556
Please note while programming:.....	556
Cycle parameters.....	556

21 Touch probe cycles.....	557
21.1 General information about touch probe cycles.....	558
Method of function.....	558
Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes.....	558
21.2 Before You Start Working with Touch Probe Cycles.....	559
Maximum traverse to touch point: DIST in touch probe table.....	559
Set-up clearance to touch point: SET_UP in touch probe table.....	559
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table.....	559
Touch trigger probe, probing feed rate: F in touch probe table.....	560
Touch trigger probe, rapid traverse for positioning: FMAX.....	560
Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table.....	560
Executing touch probe cycles.....	561
21.3 Touch probe table.....	562
General information.....	562
Editing touch probe tables.....	562
touch probe data.....	563
21.4 Fundamentals.....	564
Overview.....	564
Setting machine parameters.....	566
Entries in the tool table TOOL.T.....	568
21.5 Calibrate the TT (Cycle 480, Option 17).....	570
Cycle run.....	570
Please note while programming:.....	570
Cycle parameters.....	570
21.6 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, Option 17).....	571
Fundamentals.....	571
Cycle run.....	571
Please note while programming:.....	572
Cycle parameters.....	572
21.7 Measuring tool length (Cycle 481, Option 17).....	573
Cycle run.....	573
Please note while programming:.....	574
Cycle parameters.....	574

21.8 Measuring tool radius (Cycle 482, Option 17).....575

 Cycle run..... 575

 Please note while programming:.....575

 Cycle parameters..... 576

21.9 Measuring tool length and radius (Cycle 483, Option 17)..... 577

 Cycle run..... 577

 Please note while programming:.....577

 Cycle parameters..... 578

22 Tables and Overviews.....	579
22.1 Machine-specific user parameters.....	580
Application.....	580
22.2 Connector pin layout and connection cables for data interfaces.....	592
RS-232-C/V.24 interface for HEIDENHAIN devices.....	592
Non-HEIDENHAIN devices.....	594
Ethernet interface RJ45 socket.....	595
22.3 Technical Information.....	596
Technical Information.....	596
User functions.....	597
Software options.....	600
Accessories.....	600
Fixed cycles.....	601
Miscellaneous functions.....	602

1

**First Steps with
the TNC 128**

1 First Steps with the TNC 128

1.1 Overview

1.1 Overview

This chapter is intended to help TNC beginners quickly learn to handle the most important procedures. 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



Refer to your machine manual.

Danger exists for the operator when the machine is started up. Read the safety information before switching on the machine.



Switch-on and crossing over the reference points can vary depending on the machine tool. Refer to your machine manual.

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

CE

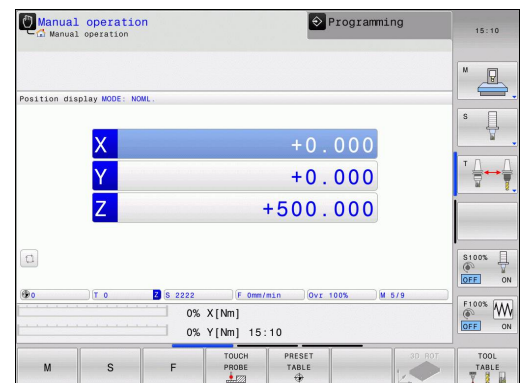
- ▶ Press the **CE** key: The TNC compiles the PLC program

I

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



- ▶ Cross the datums 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 TNC is now ready for operation in the **Manual operation** mode.

Further information on this topic

- Approaching datums
Further Information: "Switch-on", page 334
- 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 Programming operating mode key for the TNC to switch to **Programming**

Further information on this topic

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

The most important TNC keys

Key	Functions for conversational guidance
	Confirm entry and activate the next dialog prompt
	Ignore the dialog question
	End the dialog immediately
	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 a program", page 119
- Overview of keys
Further Information: "Controls of the TNC", page 2

First Steps with the TNC 128

1.3 Programming the first part

Opening a new program/file management

PGM
MGT

- ▶ Press the **PGM MGT** key: The TNC opens the file manager. The file management of the TNC is arranged much like the file management on a PC with the Windows Explorer. The file management enables you to manage data on the internal memory of the TNC.

GOTO
□

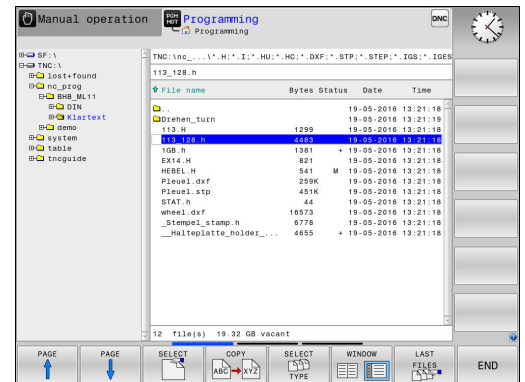
- ▶ Use the arrow keys to select the folder in which you want to open the new file.
- ▶ Press the **GOTO** key: The TNC opens a keyboard in the pop-up window.
- ▶ Enter any desired file name with the extension **.H**.

ENT

- ▶ Confirm with the **ENT** key: The control asks you for the unit of measurement for the new program.

MM

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



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

Further information on this topic

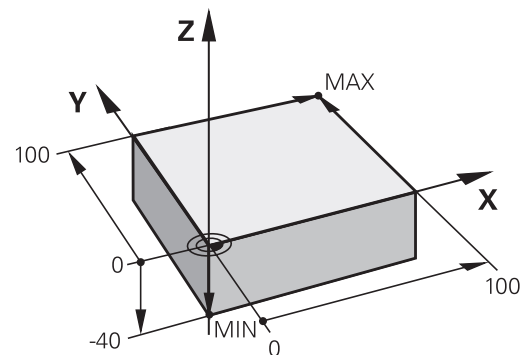
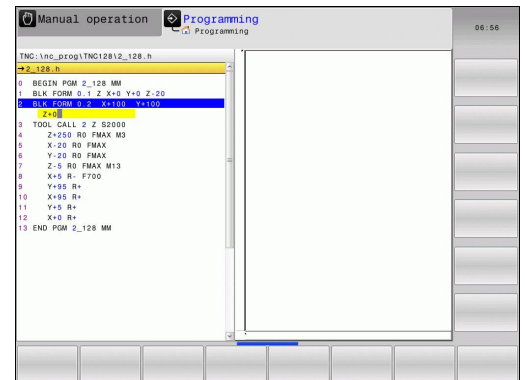
- File management
Further Information: "Working with the file manager", page 127
- Creating a new program
Further Information: "Opening programs and entering", page 112

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 reference point.

After you have selected the desired blank form via soft key, the TNC 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 reference point, e.g. 0, confirm with the **ENT** key
- ▶ **Workpiece blank def.: Minimum Y:** Smallest Y coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the **ENT** key
- ▶ **Workpiece blank def.: Minimum Z:** Smallest Z coordinate of the workpiece blank with respect to the reference point, 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 reference point, 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 reference point, 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 reference point, e.g. 0. Confirm with the **ENT** key. The TNC concludes the dialog



Example NC blocks

```
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: "Opening a new part program", page 115

First Steps with the TNC 128

1.3 Programming the first part

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

- 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: "Tool movements in the program", page 202

Recommended program layout for simple cycle programs

- 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 451

Layout of contour machining programs

```

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

```

Cycle program layout

```

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

```

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 TNC 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

- ▶ **Tool radius comp: R+/R-/no comp?** confirm 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 TNC saves the input 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

- ▶ **Tool radius comp: R+/R-/no comp?** confirm 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 TNC saves the input 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

- ▶ **Tool radius comp: R+/R-/no comp?** confirm 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 TNC saves the input positioning block

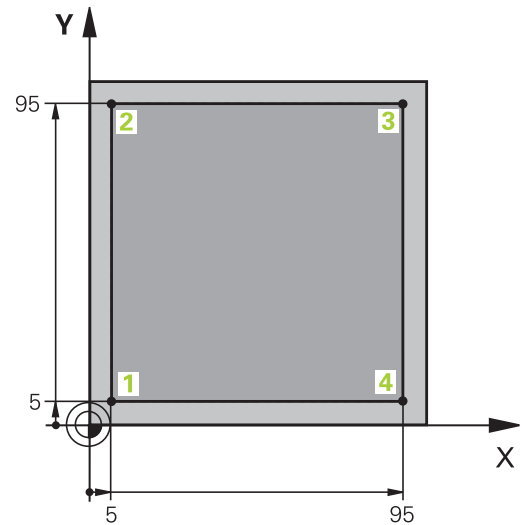
Z

- ▶ 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

- ▶ **Tool radius comp: R+/R-/no comp?** confirm 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**. Confirm with the **END** key: The TNC saves the entered positioning block



First Steps with the TNC 128

1.3 Programming the first part

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

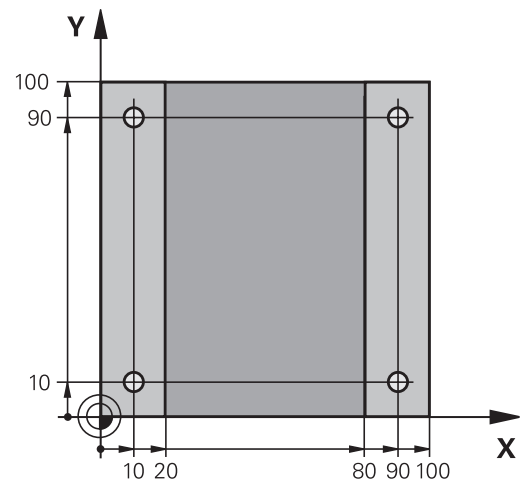
- ▶ Retract tool: Press the orange axis key Z to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- ▶ **Tool radius comp: R+/R-/no comp?** confirm with the **ENT** key: Do not activate radius compensation
- ▶ **Vorschub F=?** confirm with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

Further information on this topic

- Creating a new program
Further Information: "Opening programs and entering", 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 199
- Miscellaneous functions M
Further Information: "Miscellaneous functions for program run inspection, spindle and coolant ", page 298

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 TNC saves the entered positioning block
- ▶ Call the cycle menu: Press the CYCL DEF key

CYCL
DEF

- ▶ Display the drilling cycles

DRILLING/
THREAD

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

200

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

SPEC
FCT

- ▶ Display the functions for point machining

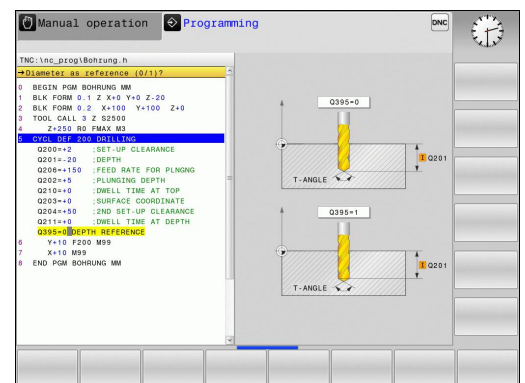
CONTOUR
+ POINT
MACHINING

- ▶ Select the pattern definition

PATTERN
DEF

- ▶ 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

POINT



First Steps with the TNC 128

1.3 Programming the first part

CYCL
CALL

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

CYCLE
CALL
PRT

- ▶ 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**. Confirm with the **END** key: The TNC saves 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 and confirm with the **END** key: The TNC saves the entered positioning block

Example NC blocks

0 BEGIN PGM C200 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 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: "Opening programs and entering", page 112
- Cycle programming
Further Information: "Fundamentals / Overviews", page 451

First Steps with the TNC 128

1.4 Graphically testing the first part

1.4 Graphically testing the first part

Selecting the correct operating mode

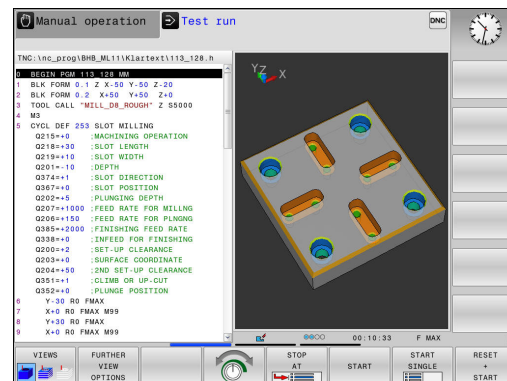
You can test programs in the **Test run** mode:



- ▶ Press the operating mode key for the TNC to switch to **Test Run** operating mode

Further information on this topic

- Operating modes of the TNC
Further Information: "Modes of operation", page 83
- Testing programs
Further Information: "Test run", page 401



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.



- ▶ Press the **PGM MGT** key: The TNC opens the file manager



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



- ▶ Press the **DEFAULT** soft key: The TNC 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 the test run



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

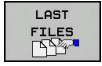
Further information on this topic

- Tool management
Further Information: "Enter tool data into the table", page 182
- Testing programs
Further Information: "Test run", page 401

Choosing the program you want to test



- ▶ Press the **PGM MGT** key: The TNC opens the file manager



- ▶ Press the **LAST FILES** soft key: The TNC 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 127

Selecting the screen layout and the view






- ▶ Press the key for selecting the screen layout. The TNC shows all available alternatives in the soft-key row



- ▶ Press the **PROGRAM + GRAPHICS** soft key: In the left half of the screen the TNC shows the program; in the right half it shows the workpiece blank

The TNC features the following views:

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

Further information on this topic

- Graphic functions
Further Information: "Graphics ", page 390
- Performing a test run
Further Information: "Test run", page 401

First Steps with the TNC 128

1.4 Graphically testing the first part

Starting the test run



- ▶ 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



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



- ▶ 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 401
- Graphic functions
Further Information: "Graphics ", page 390
- Adjusting the simulation speed
Further Information: "Speed of the setting test runs", page 391

1.5 Setting up tools

Selecting the correct operating mode

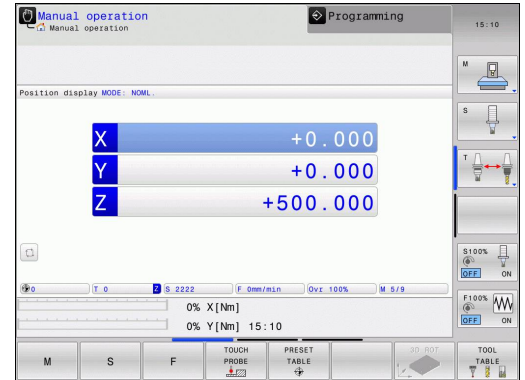
Tools are set up in the **Manual operation** mode:



- ▶ Press the operating mode key for the TNC to switch to **Manual operation** operating mode

Further information on this topic

- Operating modes of the TNC
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

First Steps with the TNC 128

1.5 Setting up tools

The tool table TOOL.T

In the tool table TOOL.T (permanently saved under **TNC:\table**), save the tool data such as length and radius, but also further tool-specific information that the TNC needs to perform its functions.

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



- ▶ Display the tool table: The TNC 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

T	NAME	L	R	R2	DL
0	NULLWerkzeug	0	0	0	0
1 D2		30	1	0	
2 D4		40	2	0	
3 D6		50	3	0	
4 D8		60	4	0	
5 D10		80	5	0	
6 D12		85	6	0	
7 D14		70	7	0	
8 D16		80	8	0	
9 D18		90	9	0	
10 D20		90	10	0	
11 D22		90	11	0	
12 D24		90	12	0	
13 D26		90	13	0	
14 D28		100	14	0	
15 D30		100	15	0	
16 D32		100	16	0	
17 D34		100	17	0	
18 D36		100	18	0	
19 D38		100	19	0	

Further information on this topic

- Operating modes of the TNC
Further Information: "Modes of operation", page 83
- Working with the tool table
Further Information: "Enter tool data into the table", page 182

The pocket table TOOL_PTCH

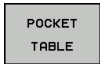


The function of the pocket table depends on the machine. Refer to your machine manual.

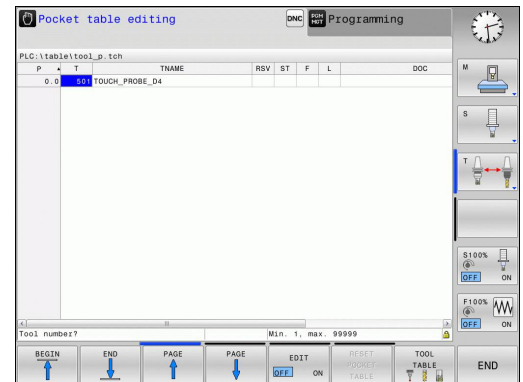
In the pocket table TOOL_PTCH (permanently saved under **TNC:\table**) you specify which tools your tool magazine contains. To enter data in the pocket table TOOL_PTCH, proceed as follows:



- ▶ Display the tool table: The TNC shows the tool table



- ▶ Display the pocket table: The TNC 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 TNC
Further Information: "Modes of operation", page 83
- Working with the pocket table
Further Information: "Pocket table for tool changer", page 190

First Steps with the TNC 128

1.6 Workpiece setup

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 for the TNC to switch to **Manual operation** operating mode **Manual operation**

Further information on this topic

- The operating mode **Manual operation**
Further Information: "Moving the machine axes", page 336

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

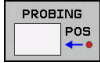
- Setting datums with a 3-D touch probe
Further Information: "Datum setting with a 3-D touch probe (option number 17)", page 375
- Setting datums without 3-D touch probe
Further Information: "Datum setting without a 3-D touch probe", page 359

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

- Insert a 3-D touch probe: In the **Positioning with manl.data input** mode, run a **TOOL CALL** block containing the tool axis and then return to the **Manual operation** mode



- Press the probing function soft key: The TNC displays all available functions in the soft key row



- Select the function for setting a datum, e.g. press the **PROBE POSITION** soft key
- Position the touch probe near the first touch point on the first workpiece edge
- Select the probing direction via soft key
- Press **NC START**: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point

Then the TNC displays the coordinates of the measured position



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

- Repeat this procedure for all axes, in which you want to set the datum

Further information on this topic

- Setting datums
Further Information: "Datum setting with a 3-D touch probe (option number 17)", page 375

First Steps with the TNC 128

1.7 Running the first program

1.7 Running the first program

Selecting the correct operating mode

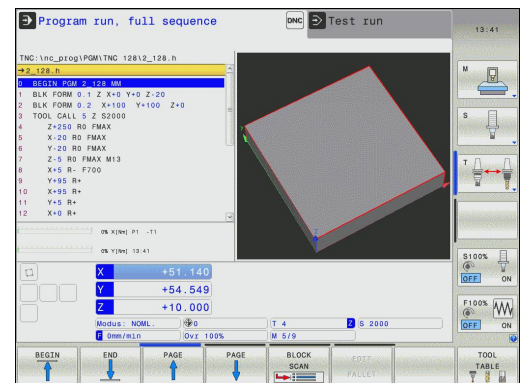
You can run programs either in the **Program run, single block** or the **Program run, full sequence** mode:



- ▶ Press the operating mode key: The TNC changes to operating mode **Program run, single block**, and the TNC executes the NC program block by block. You have to confirm each block with the **NC START KEY**



- ▶ Press the operating mode key: The TNC changes to operating mode **Program run, full sequence**, and the TNC runs 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 TNC
Further Information: "Modes of operation", page 83
- Executing a program
Further Information: "Program run", page 405

Choosing the program you want to run



- ▶ Press the **PGM MGT** key: The TNC opens the file manager



- ▶ Press the **LAST FILES** soft key: The TNC 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 127

Start the program



- ▶ Press the **NC START** key: The TNC runs the active program

Further information on this topic

- Executing a program
Further Information: "Program run", page 405

2

Introduction

Introduction

2.1 The TNC 128

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

1 Header

When the TNC is on, the screen displays the selected operating modes in the header: The machining mode at the 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 TNC messages also appear.

2 Soft keys

In the footer the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The 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 highlighted

3 Soft-key selection keys

4 Keys for switching the soft keys

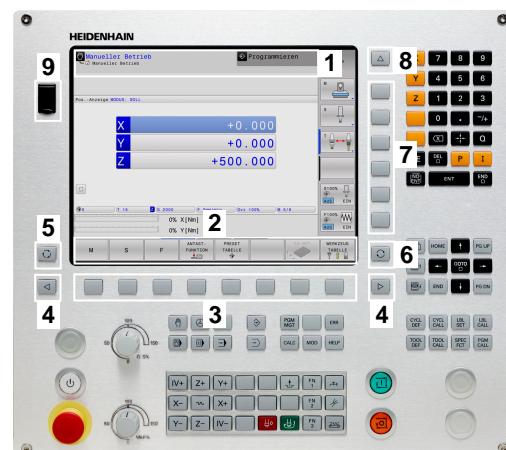
5 Setting the screen layout

6 Shift key for switchover between machining and programming modes

7 Soft-key selection keys for machine tool builders

8 Keys for switching the soft keys for machine tool builders

9 USB connection



Set screen layout

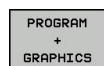
You select the screen layout yourself: In the **Programming** mode, for example, you can have the TNC show program blocks in the left window while the right window displays programming graphics. You could also display the program structure in the right window instead, or display only program blocks in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

Set up screen layout:



- Press the screen switchover key: The soft key row shows the available layout options

Further Information: "Modes of operation", page 83



- Select the desired screen layout with a soft key

Introduction

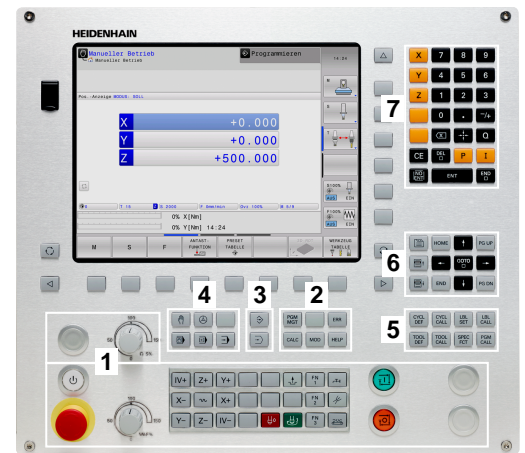
2.2 Visual display unit and operating panel

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
- 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.



Some machine tool builders do not use the standard HEIDENHAIN operating panel. Refer to your machine manual.

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



2.3 Modes of operation

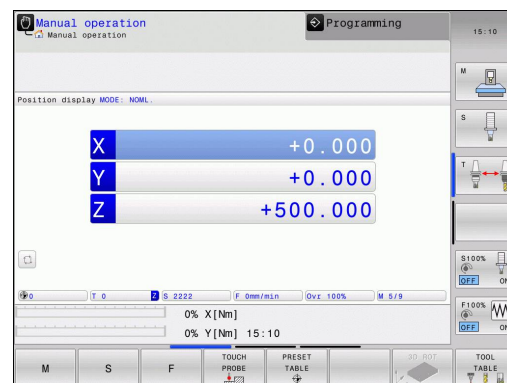
Manual Operation and El. Handwheel

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 datums.

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
	Positions
	Left: positions, right: status display

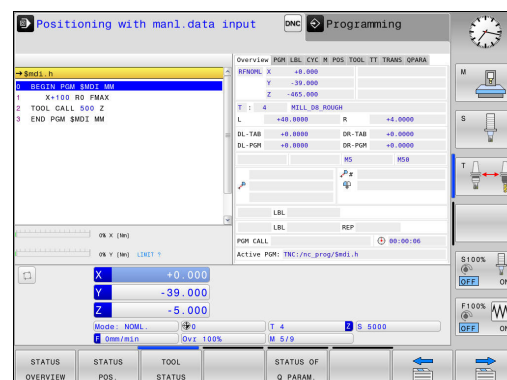


Positioning with Manual Data Input

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

Soft keys for selecting the screen layout




Soft key	Window
	Program
	Left: program, right: status display

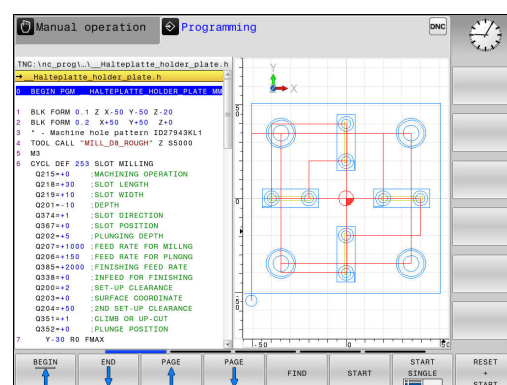


Programming

In this operating mode you can write your machining 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
	Program
	Left: program, right: program structure
	Left: program blocks, right: programming graphics



Introduction

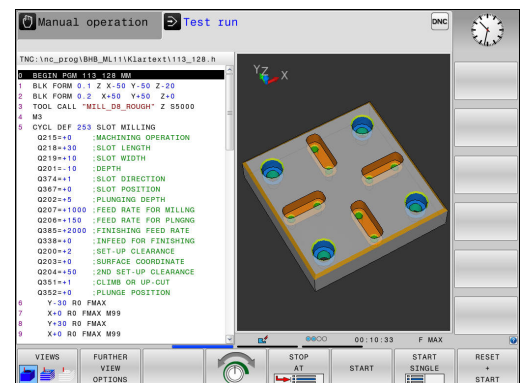
2.3 Modes of operation

Test Run

In the **Test run** mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the program or violations of the 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



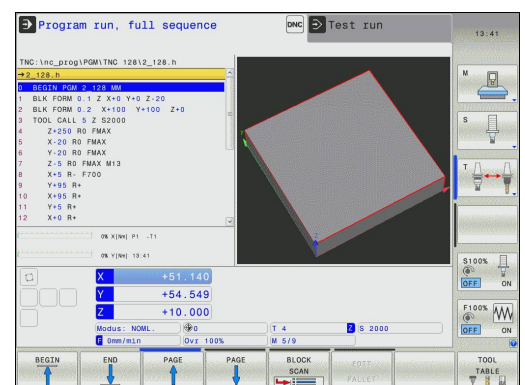
Program Run, Full Sequence and Program Run, Single Block

In the **Program run full sequence** mode, the TNC 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 with manl.data input

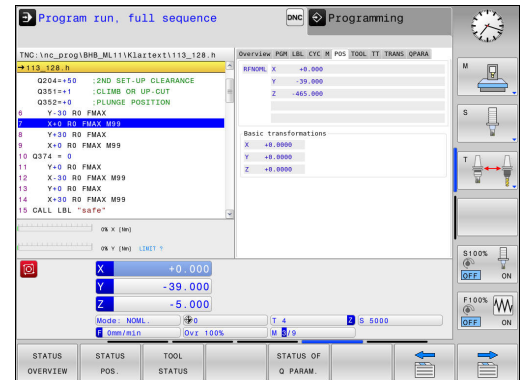








If the **GRAPHICS** screen layout is selected the status display is not shown.

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
XYZ	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information
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



Icon	Meaning
	<p>No program selected, program reselected, program aborted via internal stop or program terminated</p> <p>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.</p>
	<p>Program started, execution runs</p> <p>For safety reasons, the control permits no handling in this condition</p>
	<p>Program stopped, e.g. in operating mode</p> <p>Program run, full sequence after pressing the NC STOP key</p> <p>For safety reasons, the control permits no handling in this condition</p>
	<p>Program interrupted, e.g. in operating mode</p> <p>Positioning with manl.data input following the error-free execution of an NC block</p> <p>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!</p> <p>Further Information: "Programming and executing simple machining operations", page 384 and "Program-controlled interruptions", page 406</p>
	<p>Program aborted or terminated</p>
	<p>Pulsing spindle speed function is active</p>

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

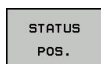


- Select the screen layout with additional status display: In the right half of the screen, the TNC shows the **OVERVIEW** status form

To select an additional status display



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



- 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



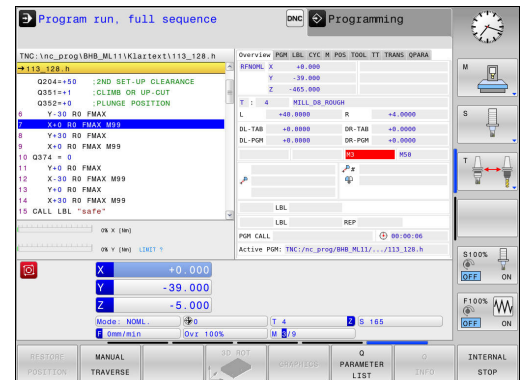
Please note that some of the status information described below is not available unless the associated software option is enabled on your TNC.

2.4 Status displays

Overview

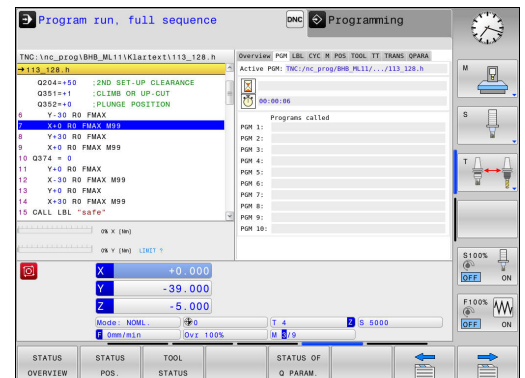
The **Overview** status form is displayed by the TNC following TNC switch-on if you selected the screen layout **PROGRAM + STATUS** (or **POSITION + STATUS**). The overview form contains a summary of the most important status information, which you can also find on the various detail forms.

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)

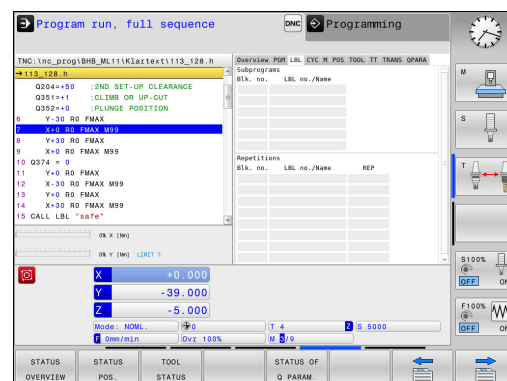
Soft key	Meaning
No direct selection possible	Name and path of the active main program
	Dwell time counter
	Machining time
	Active programs



Status displays 2.4

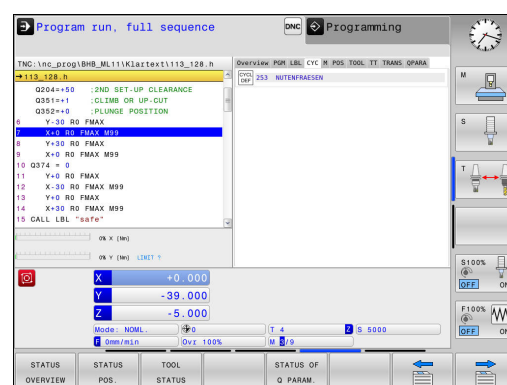
Program section repeat/Subprograms (LBL tab)

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



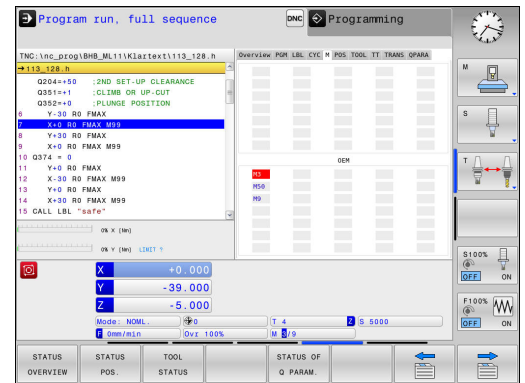
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active fixed cycle



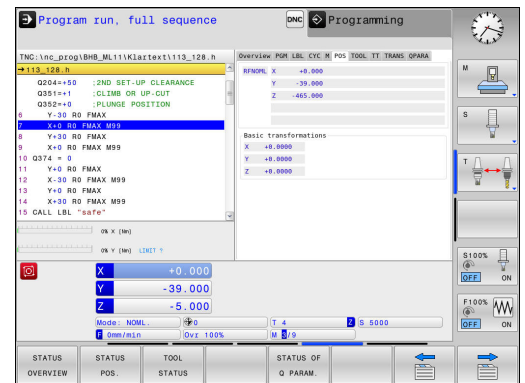
Active miscellaneous functions M (M tab)

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



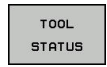
Positions and coordinates (POS tab)

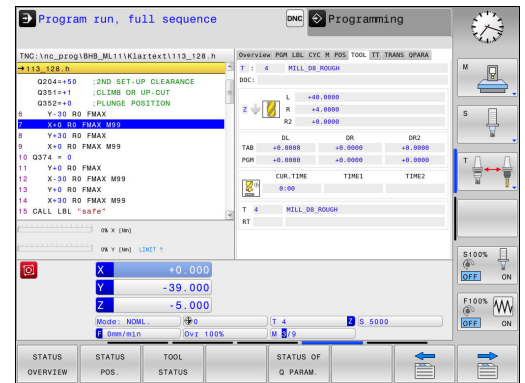
Soft key	Meaning
STATUS POS.	Type of position display, e.g. actual position



Information on tools (TOOL tab)

Soft key Meaning

	Display of active tool: <ul style="list-style-type: none"> ■ 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



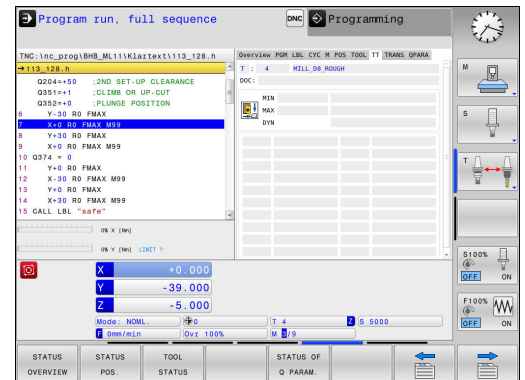
Tool measurement (TT tab)



The TNC displays the TT 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 TNC displays an active datum shift in up to 3 (5) axes
	Mirrored axes (Cycle 8)
	Active scaling factor/factors (Cycles 11 / 26); The TNC displays an active scaling factor in up to 6 axes
	Scaling datum

Cycles for coordinate transformation

Further Information: "Cycles: Coordinate Transformations", page 535

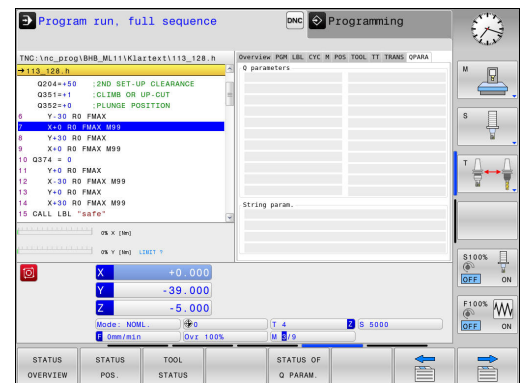
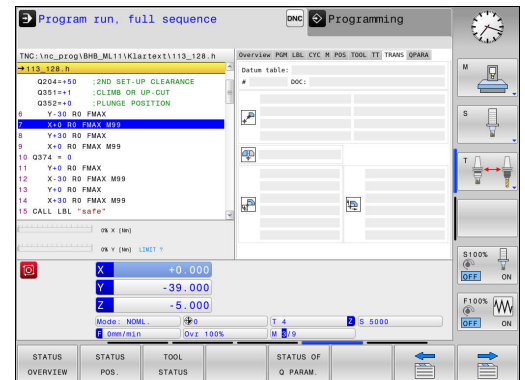
Displaying Q parameters (QPARA tab)

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 TNC 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



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

The TNC 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 TNC 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 the task bar

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: CAD viewer 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**.

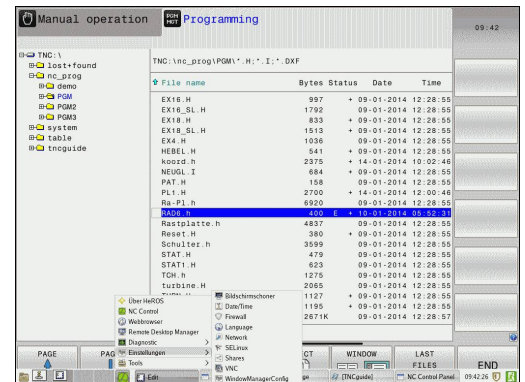


You can randomly move all open applications to the right of the green HEIDENHAIN symbol between the workspaces by pressing and holding the left mouse button.

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 96
 - **Portscan OEM:** Available only to authorized specialists
 - **RemoteService:** Start and stop remote maintenance
Further Information: "Remote Service", page 97
 - **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
 - **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 443
- **HePacketManager**: Available only to authorized specialists
- **HePacketManager Custom**: Available only to authorized specialists
- **Tools**: File applications
 - **Document Viewer**: Display files, e.g. PDF files
 - **File Manager**: Available only to authorized specialists
 - **Geeqie**: Open and manage graphics
 - **Gnumeric**: Open and edit tables
 - **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 139

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 contains the type of port (TCP/UDP), the port number, the offering 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:

- ▶ Open the task bar at the bottom edge of the screen
Further Information: "Window manager", page 93
- ▶ 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:

- ▶ Open the task bar at the bottom edge of the screen
Further Information: "Window manager", page 93
- ▶ 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 TNC", page 437

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.

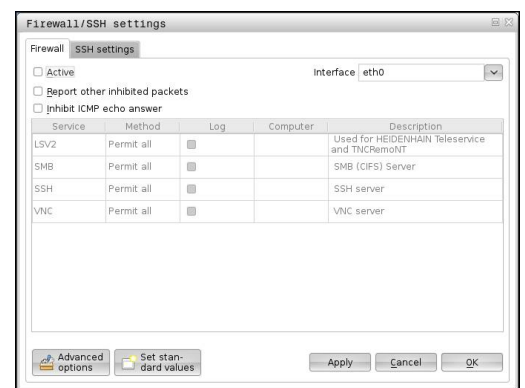
Setting up the control

To set up the control, proceed as follows:

- ▶ Open the task bar at the bottom edge of the screen
Further Information: "Window manager", page 93
- ▶ 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** key to save the settings
- ▶ Press the **OK** button
- ▶ The firewall is disabled.



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.

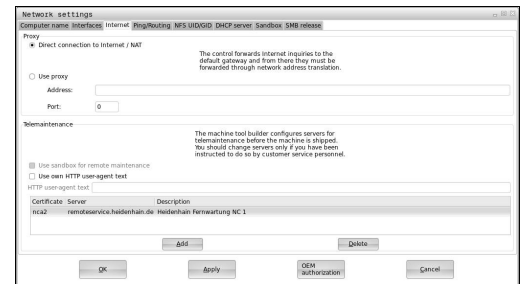
To install the certificate on the control proceed as follows:

- ▶ Open the task bar at the bottom edge of the screen
Further Information: "Window manager", page 93
- ▶ 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 selection 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



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 each action must be specifically permitted otherwise the TNC will not run it. 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 TNC is prepared to permit running of only those 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 TNC runs only those applications 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:

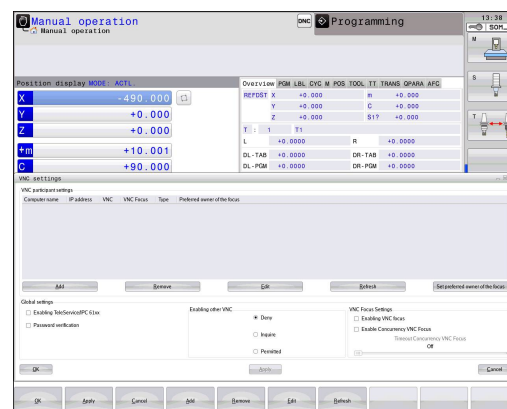
- List of permitted clients (IP address or name)
- Password for the connection
- Additional server options
- Additional settings for assigning the focus



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 to the TNC by your machine manufacturer.

Refer to your machine manual.



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 93
- ▶ 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
	Model	<ul style="list-style-type: none"> ■ 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 443.
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 functions **NC/PLC Backup** and **NC/PLC Restore** you can save and restore single directories or the complete TNC drive. You can save the backup files locally on a network drive or to USB data carriers.

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.

Open 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 93
- ▶ 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
 - Backup 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
 - User 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.

Restore data



Caution: Data may be lost!

The control overwrites existing files without a confirmation prompt.

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.

Introduction

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

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

3-D touch probes

The various HEIDENHAIN 3-D touch probes enable you to:

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

The triggering touch probes TS 220 and KT 130

These touch probes are particularly effective for datum setting and workpiece measurements. The TS 220 and KT 130 touch probes transfer the triggering signals to the TNC via a cable.

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



TT 140 tool touch probe for tool measurement

The TT 140 is a triggering 3-D touch probe for tool measurement and inspection. The TNC provides three cycles for this touch probe with which you can measure the tool length and radius either with the spindle rotating or stationary. The TT 140 features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf. The triggering signal is generated by a wear-resistant and highly reliable optical switch.



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

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 HR130 and HR150 integral handwheels, HEIDENHAIN also offers the HR 410, HR 520 and HR 550FS portable handwheels.



3

**Fundamentals, file
management**

3 Fundamentals, file management

3.1 Fundamentals

3.1 Fundamentals

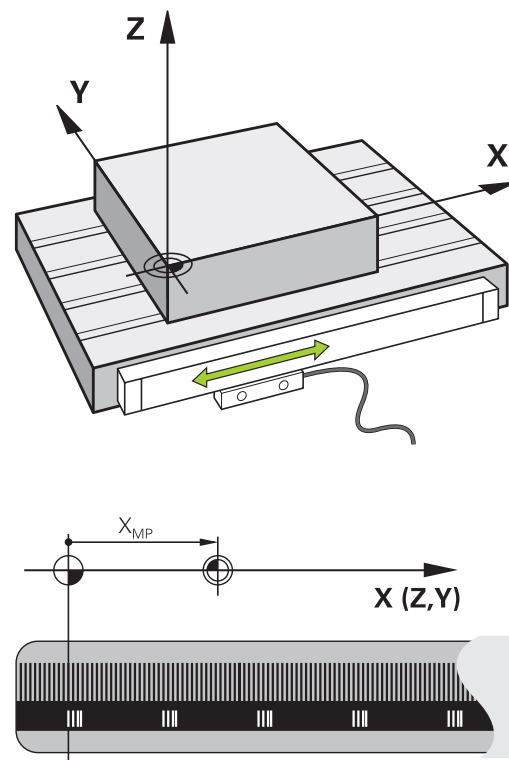
Position encoders and reference marks

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 TNC evaluates this signal and calculates the precise actual position of the machine axis.

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

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

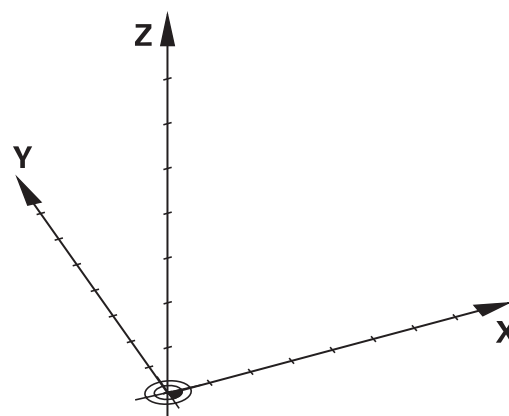


Reference system

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

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

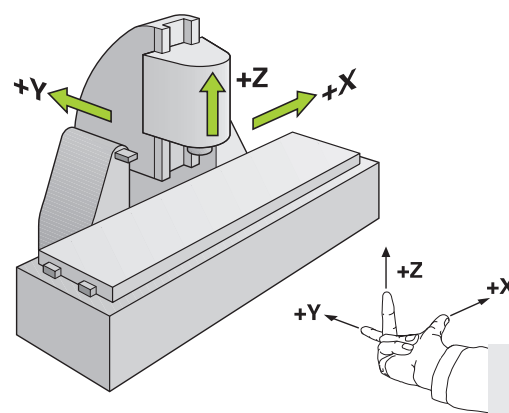
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (reference point) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.



Reference system 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

3 Fundamentals, file management

3.1 Fundamentals

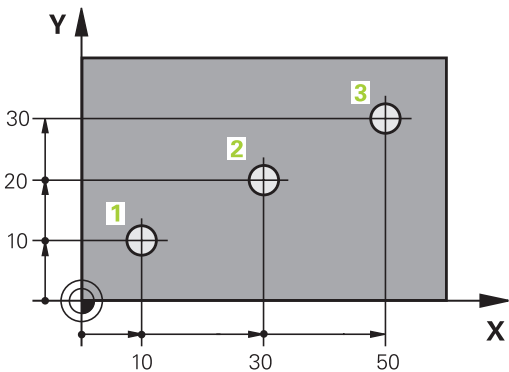
Absolute and incremental workpiece positions

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 function "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

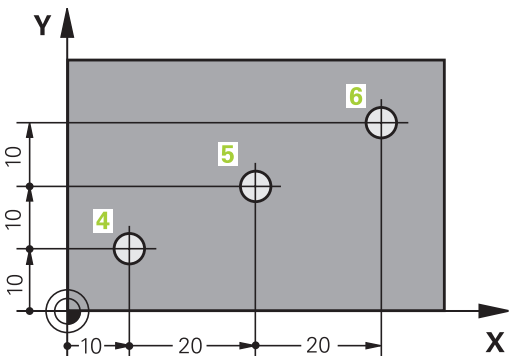
X = 20 mm

Y = 10 mm

Hole 6, with respect to 5

X = 20 mm

Y = 10 mm



Selecting the datum

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

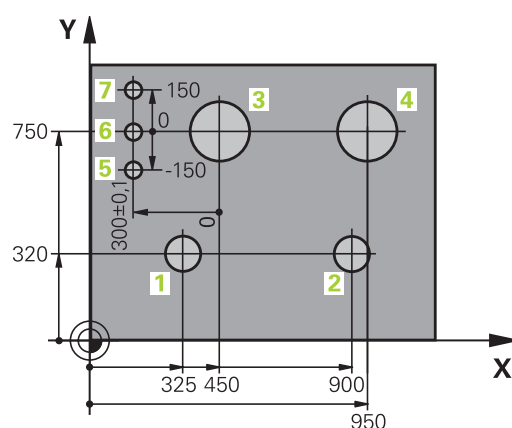
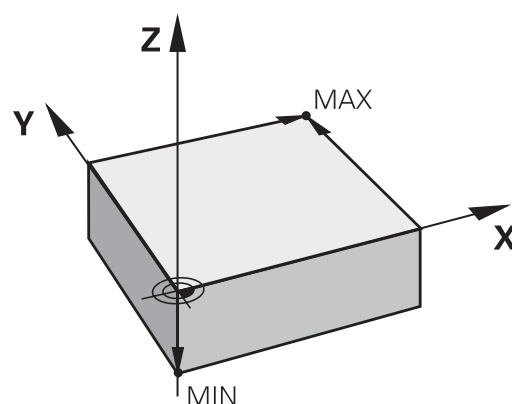
If the production drawing is dimensioned in relative datums, simply use the coordinate transformation cycles.

Further Information: "DATUM SHIFT (Cycle 7)", page 537

If the production drawing is not dimensioned for NC, set the datum 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 datum with the coordinates $X=0$ $Y=0$. The coordinates of holes 5 to 7 refer to the relative datum with the absolute coordinates $X=450$ $Y=750$. By using the **ZERO POINT DISPL.** 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 Opening programs and entering

3.2 Opening programs and entering

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 TNC numbers the blocks of a part program in ascending sequence.

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

The subsequent blocks contain information on:

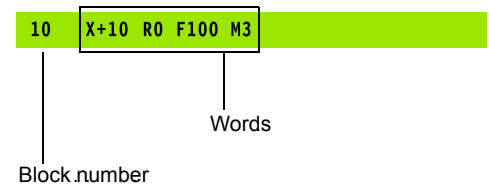
- The workpiece blank
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds, as well as
- Movements, cycles and other functions

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



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

Block





Define the blank: BLK FORM

Immediately after initiating a new program, you define a cuboid, 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 TNC 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 TNC 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: Display the BLK FORM in the NC program

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

3 Fundamentals, file management

3.2 Opening programs and entering

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: Display the BLK FORM CYLINDER in the NC program

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

Opening a new part program

You always enter a machining program in **Programming** mode. An example of program initiation:



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



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

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

FILE NAME = NEW.H



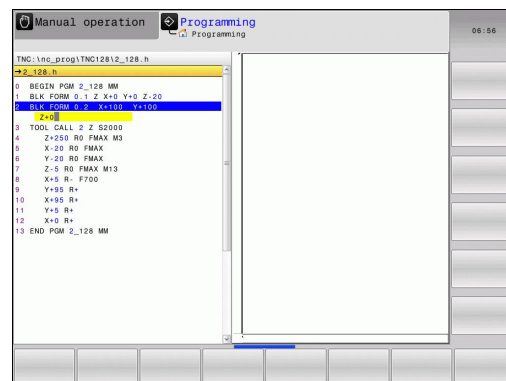
- ▶ Enter the new program name and confirm your entry with the **ENT** key



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



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



WORKING PLANE IN GRAPHIC: XY



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

WORKPIECE BLANK DEF.: MINIMUM



- ▶ 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



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

Example: Display the BLK form in the NC program

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



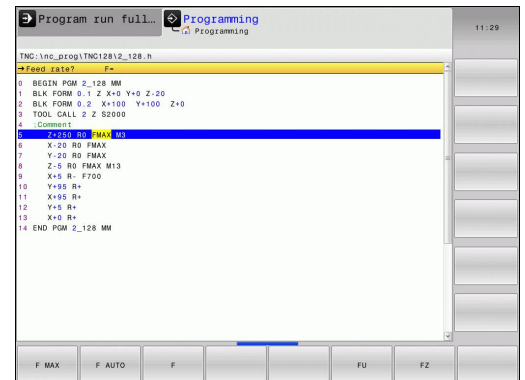
If you do not wish to define a blank form, cancel the dialog at **Working plane in graphic: XY** using the **DEL** key.

3 Fundamentals, file management

3.2 Opening programs and entering

Programming tool movements in Klartext

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



Example of a positioning block

COORDINATES ?



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



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

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



- 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)



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

MISCELLANEOUS FUNCTION M ?

- Enter **3** (miscellaneous function **M3** "Spindle ON").











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

The program-block window displays the following line:

3 X+10 R0 F100 M3

Possible feed rate input

Soft key	Functions for setting the feed rate
	Rapid traverse, blockwise
	Traverse feed rate automatically calculated in TOOL CALL
	Move at the programmed feed rate (unit of measure is mm/min or 1/10 inch/min). With rotary axes, the TNC interprets the feed rate in degrees/min, regardless of whether the program is written in mm or inches
	Define the feed per revolution (units in mm/1 or inch/1). Caution: In inch-programs, FU cannot be combined with M136
	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
	Ignore the dialog question
	End the dialog immediately
	Abort the dialog and erase the block

3.2 Opening programs and entering

Actual position capture

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

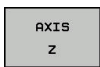
- Positioning-block programming
- Cycle programming

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 TNC displays the axes whose positions can be transferred



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



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

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












The TNC keeps the soft-key row for axis selection active until you deactivate it by pressing the actual-position-capture key again. This behavior remains in effect even if you save the current block and open a new one with a axis key. If you select a block element in which you must choose an input alternative via soft key (e.g. for radius compensation), then the TNC also closes the soft-key row for axis selection.

Editing a program



You cannot edit a program while it is being run by the TNC in a machine operating mode.




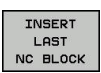
While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys or the soft keys:

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
	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
	Move from one block to the next
	
	Select individual words in a block
	
	<p>To select a certain block, press the GOTO key, enter the desired block number, and confirm with the ENT key.</p> <p>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</p>

3

Fundamentals, file management

3.2 Opening programs and entering

Soft key/key	Function
	<ul style="list-style-type: none"> ■ Set the selected word to zero ■ Erase an incorrect number ■ Delete the (clearable) error message
	Delete the selected word
	<ul style="list-style-type: none"> ■ Delete the selected block ■ Erase cycles and program sections
	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

Save 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
 - ▶ Press the **STORE** soft key for the TNC to save 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
 - ▶ Press the **SAVE AS** soft key: The TNC 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

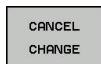


The file saved with **SAVE AS** is now also found in the file management under **LAST FILES**.

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



- ▶ Press the **CANCEL CHANGE** soft key: The TNC opens a window in which you can confirm or cancel this action
- ▶ Confirm with the **YES** key or cancel with the **ENT** key, or press the **NO** 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 have started a search in a very long program, the TNC shows a progress display window. You then have the option of canceling the search via soft key.

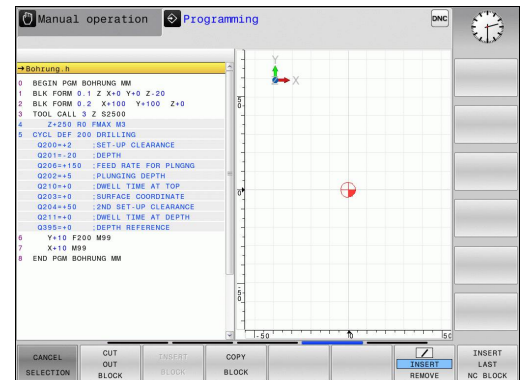
3 Fundamentals, file management

3.2 Opening programs and entering

Marking, copying, cutting and inserting program sections

The TNC 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 TNC 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 TNC shows the marked blocks in a different color. You can end the marking function at any time by pressing the **CANCEL SELECTION** soft key
- ▶ Copy the selected program section: Press the **COPY BLOCK** soft key. Cut the selected program section: Press the **CUT OUT BLOCK** soft key. The TNC stores the selected block
- ▶ Using the arrow keys, select the block after which you wish to insert the copied (cut) program section



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

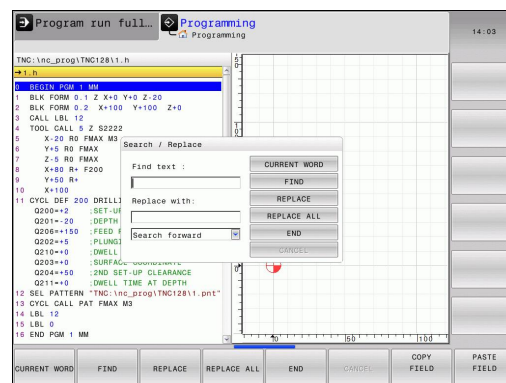
- ▶ Insert the saved program section: Press the **INSERT BLOCK** soft key
- ▶ To end the marking function, press the **CANCEL SELECTION** soft key

The TNC search function

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

Finding any text

- ▶ Select the Search function: The TNC 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
- ▶ Start the search process: The TNC moves to the next block containing the text you are searching for
- ▶ Repeat the search process: The TNC moves to the next block containing the text you are searching for
- ▶ Terminate the search function: Press the END soft key



Finding/Replacing any text



The find/replace function is not possible if

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

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

- ▶ Select the block containing the word you wish to find
- ▶ Select the Search function: The TNC superimposes the search window and displays the available search functions in the soft-key row
- ▶ Press the **CURRENT WORD** soft key: The TNC loads the first word of the current block. If required, press the info key again to load the desired word.
- ▶ Start the search process: The TNC moves to the next occurrence of the text you are searching for
- ▶ 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
- ▶ Terminate the search function: Press the END soft key

3.3 File management: Basics

3.3 File management: Basics


Files

Files in the TNC	Type
Programs	
in HEIDENHAIN format	.H
Tables for	
tools	.T
Tool changers	.TCH
Datums	.D
Points	.PNT
Reference points	.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 TNC, you must first enter a program name. The TNC saves the program to the internal memory as a file with the same name. The TNC can also save texts and tables as files.

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

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



Depending on the setting, the TNC generates a backup file (*.bak) after editing and saving of NC programs. This can reduce the memory space available to you.

File names

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

File name	File type
PROG20	.H

File names on the TNC must comply with this standard: The Open Group Base Specifications Issue 6 IEEE Std 1003.1, 2004 Edition (Posix-Standard). Accordingly, the file names may include the characters below:

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 . _ -

You should not use any other characters in file names in order to prevent any file transfer problems. Table names must start with a letter.



The maximum permitted path length is 255 characters. All drive characters, directory and the file name, including the extension, must not exceed 255 characters.

Further Information: "Paths", page 127

3.3 File management: Basics

Displaying externally generated files on the TNC

The TNC 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	pdf
Excel tables	xls
	csv
Internet files	html
Text files	txt
	ini
Graphics files	bmp
	gif
	jpg
	png

Further Information: "Additional tools for management of external file types", page 139


Data Backup

We recommend saving newly written programs and files on 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 TNC.

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 TNC 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 "\".



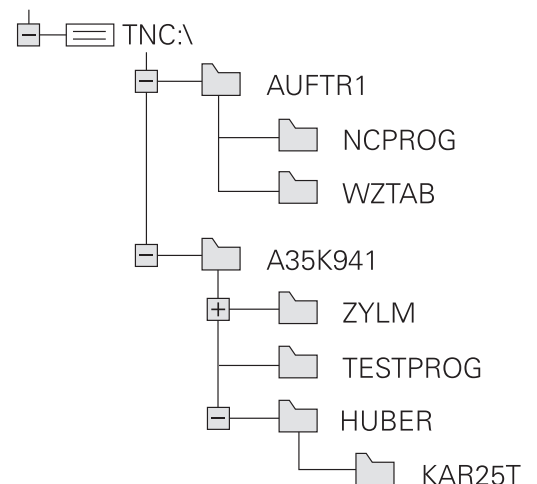
The maximum permitted path length is 255 characters. All drive characters, directory and the file name, including the extension, must not exceed 255 characters.

Example

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

TNC:\AUFTR1\NCPRG\PROG1.H




















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



3 Fundamentals, file management

3.4 Working with the file manager

Overview: Functions of the file manager

Soft key	Function	Page
	Copy a single file	132
	Display a specific file type	130
	Create new file	132
	Display the last 10 files that were selected	135
	Delete a file	136
	Tag a file	137
	Rename file	137
	Protect a file against editing and erasure	138
	Cancel file protection	138
	Import tool table of an iTNC 530	188
	Customize table view	314
	Manage network drives	147
	Select the editor	138
	Sort files by properties	138
	Copy a directory	135
	Delete directory with all its subdirectories	
	Refresh directory	
	Rename a directory	
	Create a new directory	

Calling the file manager

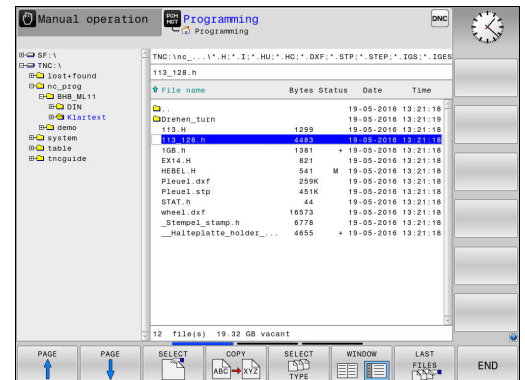


- Press the **PGM MGT** key: The TNC displays the file management window (see figure for default setting. If the TNC displays a different screen layout, press the **WINDOW** soft key.)

The narrow window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. A drive is the internal memory of the TNC. 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**.

3

Fundamentals, file management

3.4 Working with the file manager

Selecting drives, directories and files



- ▶ 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



- ▶ 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



- ▶ To select a drive, press the **SELECT** soft key, or



- ▶ 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 TNC opens the selected file in the operating mode from which you called the file manager.



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.

3

Fundamentals, file management

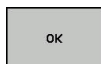
3.4 Working with the file manager

Creating a new directory

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



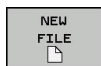
- ▶ Press the **NEW DIRECTORY** soft key
- ▶ Enter a directory name
- ▶ Press the **ENT** key



- ▶ Press the **OK** soft key to confirm or
- ▶ Press the **CANCEL** soft key to abort

Create new file

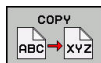
- ▶ Select the directory in the left window in which you wish to create the new file
- ▶ Position the cursor in the right window



- ▶ 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 TNC opens a pop-up window

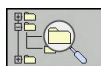


Copying files into the current directory

- ▶ Enter the name of the destination file.
- ▶ Press the **ENT** key or **OK** soft key: The TNC copies the file into the current 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 **OK** soft key: The TNC copies the file with the same name into the chosen directory. The original file is retained.



When you start the copying process with **ENT** or the **OK** soft key, the TNC 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

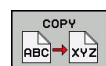
- Press the **SHOW TREE** soft key
- Select the directory with the files to copy and press the **SHOW FILES** soft key to display them



- 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: "Tag files", page 137

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

Overwriting files

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

- 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.

Fundamentals, file management

3.4 Working with the file manager

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



The **REPLACE FIELDS** function is used to overwrite lines in the target table. To avoid losing data, create a backup copy of the original table.

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 table TOOL.T using the TNC file manager. The TNC asks if you wish to overwrite the existing TOOL.T tool table:
- ▶ If you press the **REPLACE FIELDS** soft key, the TNC will completely overwrite the current TOOL.T tool table. After the copying process the new TOOL.T table consists of 10 lines.
- ▶ Or press the **REPLACE FIELDS** soft key for the TNC 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 **TAG**
- ▶ Select additional lines, if required
- ▶ Press the **SAVE AS** soft key **SAVE AS**
- ▶ 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 TNC opens the window for selecting the target directory
- ▶ Select the target directory and confirm with **ENT** or the **OK** soft key: The TNC copies the selected directory and all its subdirectories to the selected target directory

Choose one of the last files selected



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

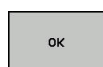


- ▶ To display the last ten files selected: press the **LAST FILES** soft key

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



- ▶ Moves the cursor up and down within a window



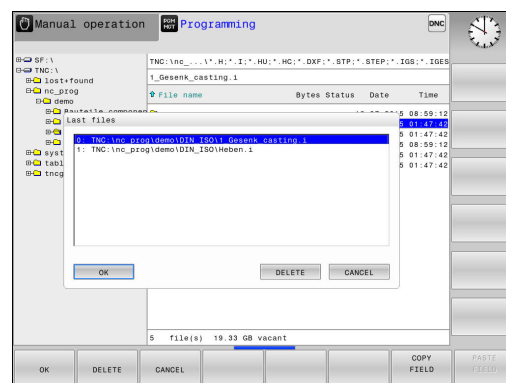
- ▶ To select the file, press the **OK** soft key, or



- ▶ 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.



3 Fundamentals, file management

3.4 Working with the file manager

Deleting a file



Caution: Data may be lost!

Once you delete files they cannot be restored!

- ▶ Move the cursor to the file you want to delete



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

Deleting a directory



Caution: Data may be lost!





Once you delete files they cannot be restored!

- ▶ Move the cursor to the directory you want to delete



- ▶ To select the erasing function, press the **DELETE** soft key. The TNC inquires whether you really intend 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

Tag 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:

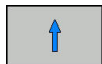
- Move the cursor to the first file



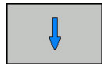
- To display the tagging functions, press the **TAG** soft key



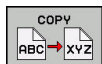
- To tag the file, press the **TAG FILE** soft key



- 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.



- Copy the tagged files: Press the **COPY** soft key; or

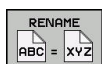


- Delete tagged files: leave active soft key row
- Press the **DELETE** soft key to delete tagged files



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

Fundamentals, file management

3.4 Working with the file manager

Sort 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



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/removing a USB device

- ▶ Move the cursor to the left-hand window
 - ▶ To select the additional functions, press the **MORE FUNCTIONS** soft key.
 - ▶ Shift the soft-key row
 - ▶ Search for a USB device
- ▶ To remove the USB device, move the cursor to the USB device in the directory tree
 - ▶ Remove the USB device



Further Information: "USB devices on the TNC", page 148

Additional tools for management of external file types

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

File types	Description
PDF files (pdf)	page 140
Excel spreadsheets (xls, csv)	page 141
Internet files (htm, html)	page 142
ZIP archives (zip)	page 143
Text files (ASCII files, e.g. txt, ini)	page 144
Video files	page 144
Graphics files (bmp, jpg, gif, png)	page 145




If you transfer files from a PC to the control by means of TNCremo, you must have entered the file name extension pdf, xls, zip, bmp gif, jpg and png in the list of the file types for binary transmission (menu item >**Extras** >**Configuration** >**Mode** in TNCremo).

3 Fundamentals, file management

3.4 Working with the file manager

Displaying PDF files

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

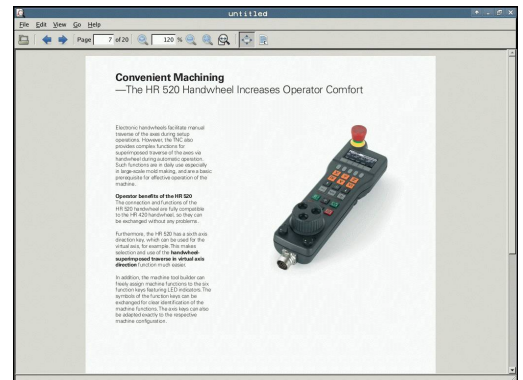
- 
 - ▶ To call the file manager, press the **PGM MGT** key.
 - ▶ Select the directory in which the PDF file is saved
 - ▶ Move the cursor to the PDF file
 - ▶ Press **ENT**: The TNC opens the PDF file in its own application using the **PDF viewer** additional tool



With the key combination ALT+TAB you can always return to the TNC 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 TNC interface.





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 the menu item **Close**: The TNC returns to the file manager



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
- 
 - ▶ Select the menu item **Close**: The TNC returns to the file manager and confirm with the **ENT** key: The TNC returns to the file manager

ENT

Displaying and editing Excel files

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

- 
 - ▶ 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
- 
 - ▶ Press **ENT**: The TNC 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 TNC 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 TNC interface.





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 the menu item **Close**: The TNC returns to the file manager

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
- 
 - ▶ Select the **Close** menu item and confirm with the **ENT** key: The TNC returns to the file manager

ENT

3 Fundamentals, file management

3.4 Working with the file manager

Displaying Internet files

To open Internet files with the extension **htm** or **html** directly on the TNC, proceed as follows:

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
- ▶ Press **ENT**: The TNC opens the Internet file in its own application using the additional **Web Browser** tool

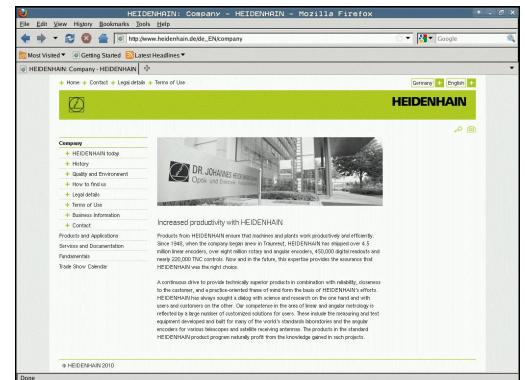
ENT



With the key combination ALT+TAB you can always return to the TNC 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 TNC interface.



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 the menu item **Quit**: The TNC returns to the file manager

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

↓

- ▶ Select the **Quit** menu item and confirm with the **ENT** key: The TNC returns to the file manager

ENT

Working with ZIP archives

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

- 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
 - ▶ Press **ENT**: The TNC opens the archive file in its own application using the additional **Xarchiver** tool
- ENT**



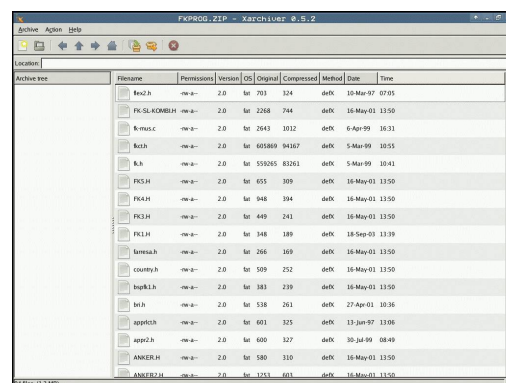
With the key combination ALT+TAB you can always return to the TNC user interface while leaving the archive file open. Alternatively, you can also click the corresponding symbol in the task bar to switch back to the TNC interface.



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**.



Please note that the TNC does not carry out any binary-to-ASCII conversion or vice versa when compressing or decompressing NC programs and NC tables. When such files are transferred to TNC controls using other software versions, the TNC may not be able to read them.



Proceed as follows to exit **Xarchiver**:

- ▶ Use the mouse to select the **ARCHIVE** menu item
- ▶ Select the menu item **Exit**: The TNC returns to the file manager

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

- ▶**
- ▶ Press the key for switching the soft keys:
The **Xarchiver** opens the **ARCHIVE** pull-down menu **ARCHIVE**
 - ▶ Select the **Exit** menu item and confirm with the **ENT** key: The TNC returns to the file manager



ENT

3

Fundamentals, file management

3.4 Working with the file manager

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
- ▶ Press the **ENT** key: The TNC opens the text file with the internal text editor

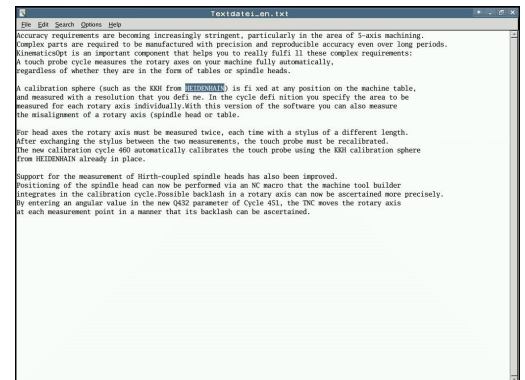
ENT



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 TNC 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 TNC interface.



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 the menu item **Exit**: The TNC returns to the file manager

Displaying video files



Refer to your machine manual.
This feature must be enabled and adapted by the machine tool builder.

Proceed as follows to open video files directly on the TNC:

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
- ▶ Press **ENT**: The TNC opens the video file in its own application

ENT

Displaying graphic files

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

- 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 TNC opens the graphics file in its own application using the additional **ristretto** tool



With the key combination ALT+TAB you can always return to the TNC 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 TNC interface.



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 the menu item **Exit**: The TNC returns to the file manager

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

- ▶**
- ▶ Press the key for switching the soft keys: The **ristretto** additional tool opens the **File** pull-down menu
- ↓**
- ▶ Select the **Exit** menu item and confirm with the **ENT** key: The TNC returns to the file manager

ENT

3 Fundamentals, file management

3.4 Working with the file manager

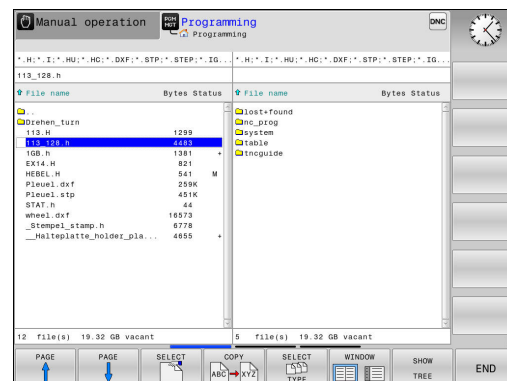
Data transfer to or from an external data carrier



Before you can transfer data to an external data medium, you must set up the data interface.

Further Information: "Setting up data interfaces", page 431

Depending on the data transfer software you use, problems can occur occasionally when you transmit data over a serial interface. They can be overcome by repeating the transmission.



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



- 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 TNC to the external data medium, move the highlight in the left window to the file to be transferred.

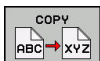
If you wish to copy from the external data medium to the TNC, move the highlight in the right window to the file to be transferred.



- Select another drive or directory: Press the **SHOW TREE** soft key



- Use the arrow keys to select the desired directory
- Select the desired file: Press the **SHOW FILES** soft key



- 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 TNC, informing about the copying progress, or



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

The TNC in a network



You must connect the Ethernet card to the network.

Further Information: "Ethernet interface ", page 437

The TNC logs error messages during network operation.

Further Information: "Ethernet interface ", page 437

If the TNC 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.

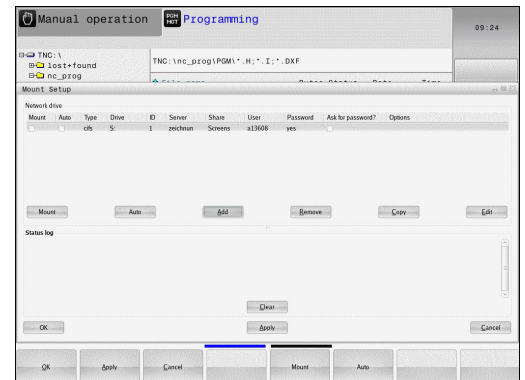
Connecting and disconnecting a network drive

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 TNC 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 TNC marks the Mount column.
Separate	End network connection
Auto	Automatically establish network connection whenever the TNC is switched on. The TNC 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

3 Fundamentals, file management

3.4 Working with the file manager

USB devices on the TNC

**Caution: Data may be lost!**

Only use the USB interface for transferring and saving, not for processing or running programs.

Backing up data from or loading onto the TNC is especially easy with USB devices. The TNC 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 TNC automatically detects these types of USB devices when connected. The TNC does not support USB devices with other file systems (such as NTFS). The TNC displays the **USB: TNC does not support device** error message when such a device is connected.



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

The TNC also displays the **USB: TNC does not support device** error message if you connect a USB hub. In this case, simply acknowledge the message with the **CE** key.

In theory, you should be able to connect all USB devices with the file systems mentioned above to the TNC. It may occur that a USB device is not correctly detected by the control. In such cases, use another USB device.

Working with USB devices











Your machine tool builder can assign permanent names for USB devices. The machine manual provides further information.

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 soft key **VERBERGEN** and file transfer is continued in the background. The control displays a warning until file transfer is completed.

Remove the USB device

Proceed as follows to remove a USB device:

- 
 - ▶ To call the file manager, press the **PGM MGT** key
- 
 - ▶ Select the left window with the arrow key
- 
 - ▶ Use the arrow keys to select the USB device to be removed
- 
 - ▶ Scroll through the soft-key row
- 
 - ▶ Press the MORE FUNCTIONS soft key
- 
 - ▶ Scroll through the soft-key row
- 
 - ▶ Select the function for removing USB devices. The TNC removes the USB device from the directory tree and reports **The USB device can be removed now.**
- 
 - ▶ Remove the USB device
 - ▶ Quit the File Manager

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

- 
 - ▶ Select the function for reconnection of USB devices

4

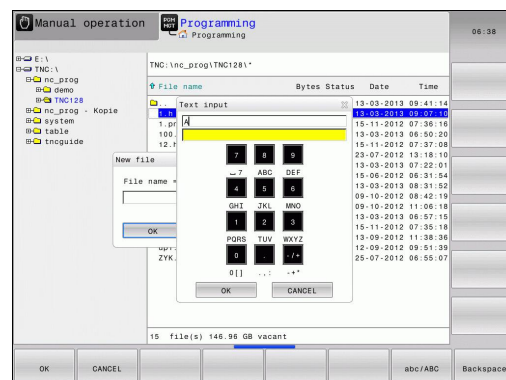
Programming aids

Programming aids

4.1 Screen keyboard

4.1 Screen keyboard

You can enter letters and special characters with the screen keypad or (if available) with a PC keyboard connected over 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 keyboard
- ▶ The TNC opens a window in which the numeric entry field of the TNC 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 TNC 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

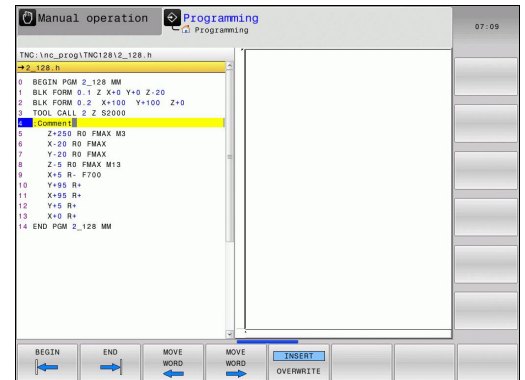
Application

You can add comments to a part program to explain program steps or make general notes.



Depending on the machine parameter **lineBreak** (no. 105404), the TNC displays comments that can no longer be shown entirely on the screen, shows these in several lines or the character >> appears on the screen.






The last character in a comment block must not have any tilde(~).



Add comments

- ▶ Select the block after which the comment is to be inserted
- ▶ Press the **SPEC FCT** key
- ▶ Press the **PROGRAMMING AIDS** soft key
- ▶ Press the **INSERT COMMENT** soft 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. Words must be separated by a space
	Jump to the end of a word. Words must be separated by a space
	Switch between paste and overwrite mode

Programming aids

4.3 Display of NC programs

4.3 Display of NC programs

Syntax highlighting

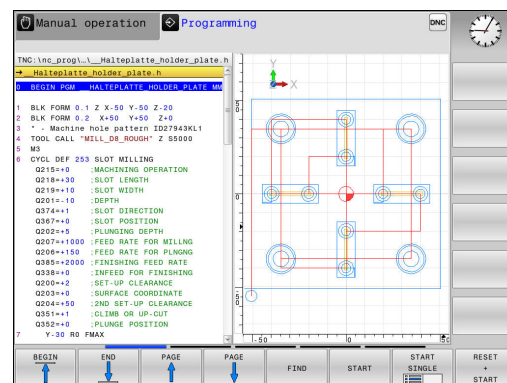
The TNC displays syntax elements with various colors according to their meaning. Programs are made more legible and clear with color-highlighting.

Color highlighting of syntax elements

Use	Color
Standard color	Black
Display of comments	Green
Display of numerical values	Blue
Block number	Violet

Scrollbar

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



4.4 Structuring programs

Definition and applications

This TNC function enables you to comment part programs in structuring blocks. Structuring blocks are short texts with up to 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 inserted structure items are managed by the TNC in a separate file (extension: .SEC.DEF). 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

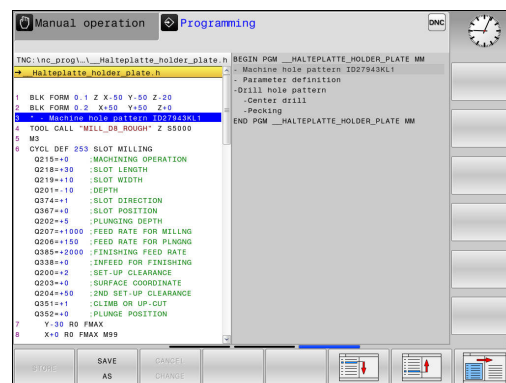
Displaying the program structure window / Changing the active window



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



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



Programming aids

4.4 Structuring programs

Inserting a structure block in the program window

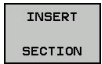
- ▶ Select the block after which the structuring block is to be inserted



- ▶ 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 TNC at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

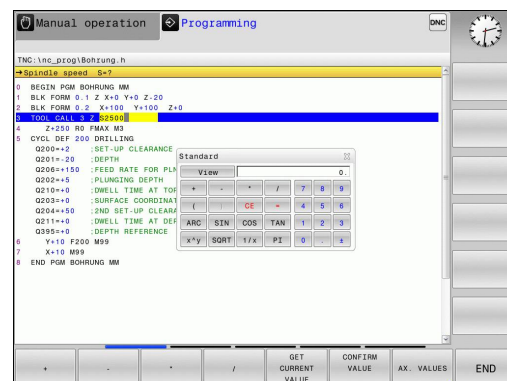
4.5 Calculator

Operation

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

- Use the **CALC** key to show and hide the on-line 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	/
Calculations in 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



Programming aids

4.5 Calculator

Calculate function	Shortcut (soft key)
Truncate decimal places	INT
Truncate integers	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 for the TNC to transfer the value into the active input box and close the calculator



You can also transfer values from a program into the calculator. When you press the **GET CURRENT VALUE** soft key or the **GOTO** key, the TNC 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
	Load the nominal or reference value of the respective axis position into the calculator
	Load the numerical value from the active input field into the calculator
	Load the numerical value from the calculator field into the active input field
	Copy the numerical value from the calculator
	Insert the copied numerical value into the 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.

Programming aids

4.6 Cutting data calculator

4.6 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 TNC 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 or spindle speed calculation:

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











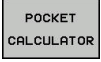



Window for feed rate calculation:

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 also calculate the feed rate in the TOOL CALL block and automatically transfer it to the subsequent positioning blocks and cycles. For this purpose, with feed rate input in positioning blocks and cycles, press the **F AUTO** soft key. The TNC then uses the feed rate defined in the TOOL CALL block. 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
	Load the spindle speed from the cutting data calculator form into an open dialog field.
	Load the feed rate from the cutting data calculator form into an open dialog field.
	Load the cutting speed from the cutting data calculator form into an open dialog field.
	Load the feed per tooth from the cutting data calculator form into an open dialog field.
	Load the feed per revolution from the cutting data calculator form into an open dialog field.
	Load the tool radius into the cutting data calculator form
	Load the spindle speed from the open dialog field into the cutting data calculator form
	Load the feed rate from the open dialog field into the cutting data calculator form
	Load the feed per tooth from the open dialog field into the cutting data calculator form
	Load the value from an open dialog field into the cutting data calculator form
	Switch to the pocket calculator
	Move the cutting data calculator in the direction of the arrow
	Use inch values in the cutting data calculator
	Close the cutting data calculator

Programming aids

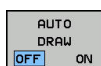
4.7 Programming graphics

4.7 Programming graphics

Generate/do not generate graphics during programming

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

- Switch the screen layout to displaying program blocks to the left and graphics to the right: Press the screen layout key and the **PROGRAM + GRAPHICS** soft key



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

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



If **AUTO DRAW** is set to **ON**, with creation of 2-D line graphics the control does not consider:

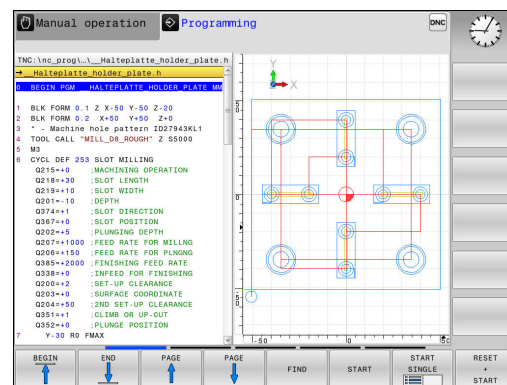
- Program section repetitions
- Jump commands
- M functions, such as M2 or M30
- Cycle calls

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

Additional functions:

Soft key	Function
	Reset previously active tool data. Generate programming graphics
	Generate programming graphic blockwise
	Generate a complete graphic or complete it after RESET + START
	Stop the programming graphics. This soft key only appears while the TNC is generating the programming graphics
	Selecting views <ul style="list-style-type: none"> ■ Plan view ■ Front view ■ Page view
	Display or hide tool paths
	Display or hide tool paths in rapid traverse

Programming aids

4.7 Programming graphics

Block number display ON/OFF



- ▶ Shift the soft-key row

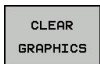


- ▶ Display block numbers: **BLOCK NO.** soft keySet **BLOCK NO. SHOW OMIT** to **SHOW**
- ▶ Hide block numbers: **BLOCK NO.** soft keySet **BLOCK NO. SHOW OMIT** to **HIDE**

Erasing the graphic



- ▶ Shift the soft-key row

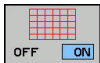


- ▶ Erase the graphics: Press the **CLEAR GRAPHICS** soft key

Showing grid lines



- ▶ Shift the soft-key row



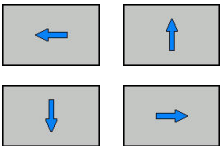

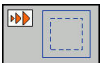

- ▶ Show grid lines: Press the **SHOW GRID LINES** soft key

Magnification or reduction of details

You can select the graphics display

- Shift the soft-key row

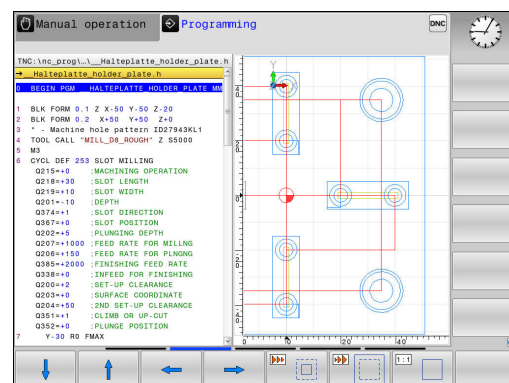
The following functions are available:

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 TNC zooms in on the defined area
- To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards



Programming aids

4.8 Error messages

4.8 Error messages

Display of errors

The TNC displays errors with e.g.:

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

When an error occurs, it is displayed in red type in the header.



The control uses different colors for different dialogs:

- 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.

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

The error message is displayed in the header until it is cleared or replaced by a higher-priority error.

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

Open the error window

ERR

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

Closing the error window

END

- Press the **END** soft key; or

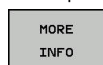
ERR

- Press the **ERR** key. The TNC closes the error window.

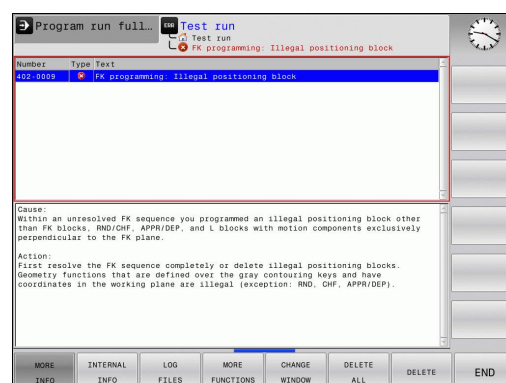
Detailed error messages

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

- Open the error window



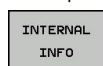
- Information on the error cause and corrective action: Position the cursor on the error message and press the **MORE INFO** soft key. The TNC opens a window with information on the error cause and corrective action.
- Leave Info: Press the **MORE INFO** soft key again



Soft key: INTERNAL INFO

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

- Open the error window

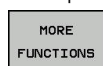


- Detailed information about the error message: Position the cursor on the error message and press the **INTERNAL INFO** soft key. The TNC opens a window with internal information about the error
- To exit Details, press the **INTERNAL INFO** soft key again

Soft key FILTER

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

Programming aids

4.8 Error messages

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 deleting the errors because the key is used for other functions.

Deleting errors

- Open the error window

DELETE

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

**DELETE
ALL**

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



If the cause of the error has not been corrected for that error it cannot be deleted. In this case, the error message remains in the window.

Error log

The TNC 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 TNC 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 TNC 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.

	► Press the LOG FILES soft key
	► Open the keystroke log file: Press the KEYSTROKE LOG soft key
	► Set the previous keystroke log if required: Press the PREVIOUS FILE soft key
	► Set the current keystroke log if required: Press the CURRENT FILE soft key

The TNC 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
	Go to beginning of keystroke log
	Go to end of keystroke log
	Find text
	Current keystroke log
	Previous keystroke log
	Up/down one line
	
	Return to main menu

Programming aids

4.8 Error messages

Informational texts

With an operating error, e.g. pressing an impermissible key or entering a value outside of a validity range, the TNC notifies you of this with an information text in the header of the operating error. The TNC deletes this information text with the next valid entry.

Save service files

If necessary, you can save the "Current status of the TNC," 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 operation).

If you repeat the **Save service files** function with the same file name, the previously saved group of service data files is overwritten. To avoid this, use another file name when you repeat the function.

Saving service files

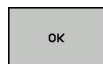
- Open the error window.



- Press the **LOG FILES** soft key



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



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

Calling the TNCguide help system

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



If your machine manufacturer also provides a help system, the TNC shows an additional **MACHINE MANUFACTURER (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.

4.9 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 175

The **TNCguide** context-sensitive help system contains the user documentation in HTML format. The TNCguide is called with the **HELP** key, and the TNC sometimes immediately displays the information specific to the situation 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 TNC always tries to start TNCguide in the language that you have selected as the conversational language on your TNC. If the files with this language are not yet available on your TNC, it automatically opens the English version.

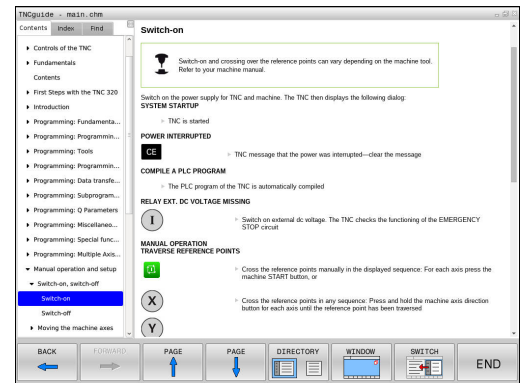
The following user documentation is available in TNCguide:

- 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.



Programming aids

4.9 TNCguide context-sensitive help system

Working with TNCguide

Call 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 TNC can open any .chm file, even if it is not saved on the TNC's internal memory



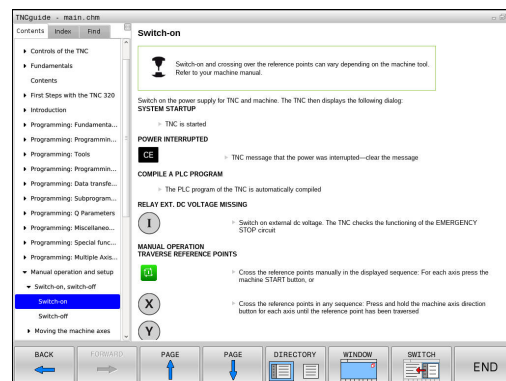
When the help system is called on the programming station, the TNC starts the internally defined standard browser.

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 TNC displays just above the soft-key row: The mouse pointer turns into a question mark
- ▶ Move the question mark to the soft key for which you want an explanation, and click: The TNC opens TNCguide. If there is no entry point for the selected soft key, then the TNC 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** button: The TNC 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
	<ul style="list-style-type: none"> If the table of contents at left is active: Select the entry above it or below it
	<ul style="list-style-type: none"> If the text window on the right is active: Move the page downward or upward if texts or graphics are not shown completely
	<ul style="list-style-type: none"> 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
	<ul style="list-style-type: none"> If the table of contents at left is active: Close the table of contents If the text window at right is active: No function
	<ul style="list-style-type: none"> 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
	<ul style="list-style-type: none"> 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
	<ul style="list-style-type: none"> If the table of contents at left is active: Select the entry above it or below it
	<ul style="list-style-type: none"> If the text window at right is active: Jump to next link
	Select the page last shown
	Page forward if you have used the "Select page last shown" function
	Move up by one page
	Move down by one page

Programming aids

4.9 TNCguide context-sensitive help system

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

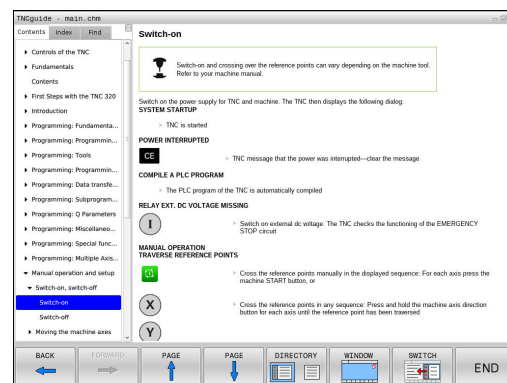
The left side is active.



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



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 desired word and confirm with the **ENT** key: The TNC lists all sources containing the word
- ▶ Use the arrow key to highlight the desired source
- ▶ Press the **ENT** key to go to the selected source



The full-text search only works for single words. If you activate the **Search only in titles** function (by mouse or by using the cursor and the space key), the TNC searches only through headings and ignores the body text.

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 TNC 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 and unpack it
- ▶ Move the unzipped CHM files to the TNC in the **TNC:\tncguide-en** directory or to the respective language subdirectory



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

Programming aids

4.9 TNCguide context-sensitive help system

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

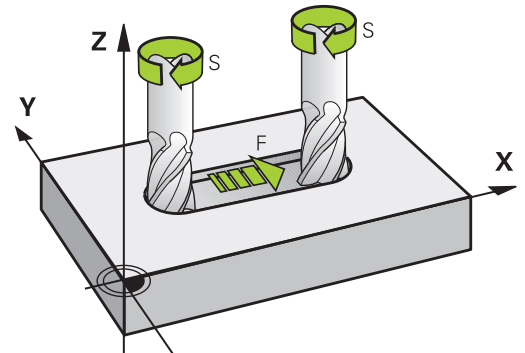
5 Tools

5.1 Entering tool-related data

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



- ▶ 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.

Changing during program run

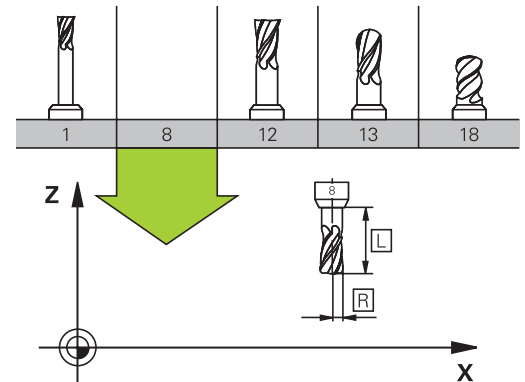
You can adjust the spindle speed during program run with the spindle speed potentiometer S.

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 TNC to calculate the tool center path i.e. the tool compensation you must also enter the length and radius of each tool you are using.

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



Tool 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

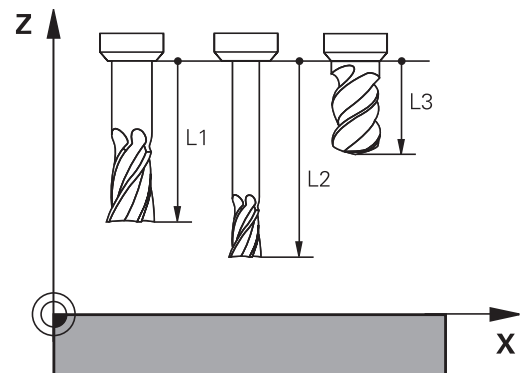
The control automatically replaces lowercase letters with corresponding uppercase letters during saving.

Impermissible characters: <Leerzeichen> ! " ' () * + ; < = > ? [/] ^ ` { } ~

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**.

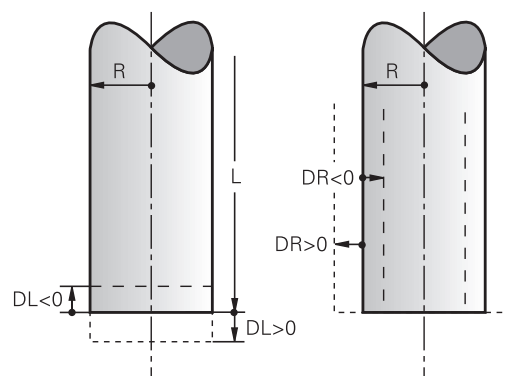
A negative delta value describes a tool undersize (**DL**, **DR**<0). An undersize is entered in the tool table for wear.

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 program



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

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 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
```

5.2 Tool data

Enter tool data into the table

You can define and store up to 767 tools and their tool data in a tool table. Also see the editing functions later in this chapter. In order to be able to assign various compensation data to a tool (indexing the tool number), insert a line and extend the tool number by a dot and a number from 1 to 9 (e.g. **T 5.2**).

You must use tool tables if:

- you wish to use indexed tools such as stepped drills with more than one length compensation value
- your machine tool has an automatic tool changer
- you want to work with Cycles 25x



if you create or manage further tool tables, the file name has to start with a letter.

You can select either list view or form view for tables via the "Screen layout" key.

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	Compensation value for tool length L	Tool length?
R	Compensation value for the 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 ToolLocked)	Tool locked? Yes=ENT/ No=NOENT
RT	Number of a replacement tool – if available – as replacement tool (RT : for ReplacementTool) 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 TNC inserts the replacement tool during the next TOOL CALL	Max. tool age for TOOL CALL?
CUR_TIME	Current age of the tool in minutes: The TNC automatically counts the current tool life (CUR_TIME : For CURrentTIME) 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 in which you can select the tool type. 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, 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	Display tool carrier kinematics using the SELECT softkey and confirm file name and path with the OK (in tool management, display using the GOTO key and confirm with the SELECT soft key). Further Information: "Allocate parameterized tool carriers", page 330	Tool-carrier kinematics
OVRTIME	Time for exceeding the tool life in minutes Further Information: "Overtime for tool life", page 195 Function is defined by the machine manufacturer. Refer to your machine manual.	Tool life expired

Tool table: Tool data required for automatic tool measurement

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 TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: length?
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: radius?
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: Radius 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 TNC 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 TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: radius?

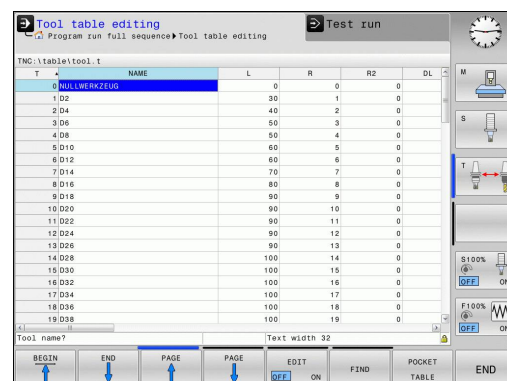
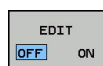
Editing the tool table

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 TNC 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 TNC shows the message: **Tool table locked**.

If a new tool is created the length and radius columns remain empty until you enter values. If it is attempted to insert such a newly created tool, the control aborts with an error message. This means you cannot insert a tool for which no data has been entered.

Displaying only specific tool types (filter setting)

- ▶ Press the **TABLE FILTER** soft key
- ▶ Select the tool type by pressing a soft key: The TNC only shows tools of the type selected
- ▶ Cancel the filter: Press the **SHOW ALL** soft key



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

5.2 Tool data

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

You can use a connected mouse or the TNC keyboard to navigate in the form. Navigation using the TNC 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 are also displayed if you navigate in the table to the right.







Opening any other tool table













- ▶ Select the **Programming** operating mode



- ▶ 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 soft key **SELECT**

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. Additional functions are illustrated in the table below.

Soft key	Editing functions for tool tables
	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
	Move to beginning of line

Soft key	Editing functions for tool tables
	Move to end of line
	Copy highlighted field
	Insert copied field
	Add the entered number of lines (tools) at the end of the table
	Adding a row with tool number for entering
	Delete the current line (tool)
	Sort the tools according to the content of a column
	Select possible entries from a pop-up window
	Show all drills in the tool table
	Show all cutters in the tool table
	Show all taps/thread cutters in the tool table
	Show all touch probes in the tool table

Exiting any other tool table

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

5.2 Tool data

Importing tool tables



Refer to your machine manual. The machine manufacturer can adapt the **IMPORT TABLE** function.

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 **IMPORT TABLE** function. The TNC 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



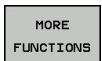
- ▶ Select the operating mode: Press the **Programming** key



- ▶ Call the file manager: Press the **PGM MGT** key



- ▶ Move the cursor to the tool table you want to import



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



- ▶ Shift the soft key row



- ▶ Press the **IMPORT TABLE** soft key: The TNC inquires whether you really want to overwrite the selected tool table

- ▶ Do not overwrite the file: Press the **CANCEL** soft key; or
- ▶ Overwrite the file: Press the **OK** soft key
- ▶ 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 _

The TNC changes a comma in the tool name to a period during import.

The TNC overwrites the selected tool table when running the **IMPORT TABLE** function. To avoid losing data, be sure to make a backup copy of your original tool table before importing it!

When tool tables are imported from an iTNC 530, all existing tools are imported along with their corresponding tool type. 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 435

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

Requirements

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

Procedure

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



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

The procedure for copying tool tables using the TNC file manager is described in the file management.

Further Information: "Copying a table", page 134

BEGIN TST .T MM			
T	NAME	L	R
1		+12.5	+9
3		+23.15	+3.5
[END]			

```

TNC640(340594) - TNCcmd
TNCcmdPlus - WIN32 Command Line Client for HEIDENHAIN Controls - Version: 5.92
Connecting with TNC640(340594) (192.168.56.101)
Connection established with TNC640, NC Software 340595 0T Dev
TNC:\nc_prog\> put tst.t tool.t /m

```

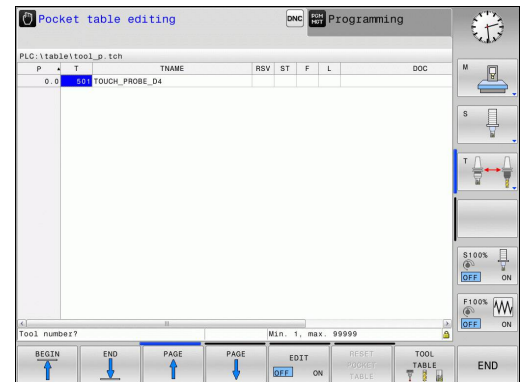
5.2 Tool data

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



- Select the pocket table: Press the **POCKET TABLE** soft key



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


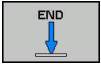


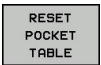
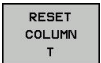
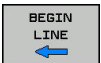
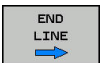




Selecting a pocket table in Programming mode



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

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 = NO ENT
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?

5.2 Tool data

Soft key	Editing functions for pocket tables
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Reset pocket table
	Reset tool number T column
	Go to beginning of the line
	Go to end of the line
	Simulate a tool change
	Select a tool from the tool table: The TNC shows the contents of the tool table. Use the arrow keys to select a tool, press OK to transfer it to the pocket table
	Edit the current field
	Sort the view



The machine manufacturer defines the features, properties and designations of the various display filters. Refer to your machine manual.

Call tool data

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

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



- **Tool number**: Enter the number or name of the tool. The tool must already be defined in a **TOOL DEF** block or in the tool table. With the **TOOL NAME** soft key you can enter a name. With the **QS** soft key you enter a string parameter. The TNC 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. If you wish to call a tool with other compensation values, also enter the index you defined in the tool table after the decimal point. There is a **SELECT** soft key for calling a window from which you can select a tool defined in the tool table TOOL.T directly without having to enter the number or name
- **Working spindle axis X/Y/Z**: Enter the tool axis
- **Spindle speed S**: Enter the spindle speed S in revolutions per minute (rpm). Instead, 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) **FU** or in millimeters per tooth (mm/tooth) **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 tool radius 2



You can also search for a tool in the pop-up window. To do so, press the **GOTO** or **FIND** soft key and enter the tool number or tool name. With the **OK** soft key you can load the tool into the dialog box.

5.2 Tool data

Example: 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.

```
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

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

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



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

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

Overtime for tool life



Refer to your machine manual.
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



The tool usage test function must be enabled by your machine manufacturer.
Refer to your machine manual.

To conduct a tool usage test, you must activate **Create tool usage files** in the MOD menu.

Further Information: "Tool usage file", page 426

Generate 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	<ul style="list-style-type: none"> ■ 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 TNC saves the path name of the corresponding NC programs. The TIME column shows the sum of all TIME entries (feed time without rapid traverse movements). The TNC sets all other columns to 0 ■ TOOLFILE: In the PATH column, the TNC saves the path name of the tool table with which you conducted the test run. This enables the TNC during the actual tool usage test to detect whether you performed the test run with the 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)

Column	Meaning
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	<ul style="list-style-type: none"> ■ 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 TNC enters the value 100 (%) during the test run
OVRMIN	Minimum feed rate override that occurred during machining. During Test Run the TNC enters the value –1
NAMEPROG	<ul style="list-style-type: none"> ■ 0: The tool number is programmed ■ 1: The tool name is programmed

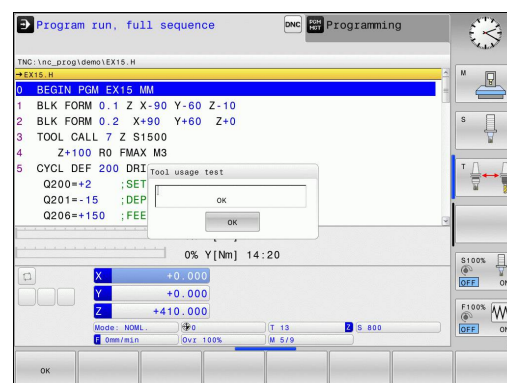
The TNC saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. This file is not visible unless the machine parameter **dependentFiles** (no. 122101) is set to **MANUAL**

Using a tool usage test

Before starting a program in the **Program Run, Full Sequence/ Single Block** operating modes, you can use the **TOOL USAGE** and **TOOL USAGE TEST** soft keys to check whether the tools being used in the selected program are available and have sufficient remaining service life. The TNC then compares the actual service life values in the tool table with the nominal values from the tool usage file.

After you have pressed the **TOOL USAGE TEST** soft key, the TNC displays the result of the tool usage test in a pop-up window. To close the pop-up window, press the **ENT** key.

You can query the tool usage test with the **FN18 ID975 NR1** function.

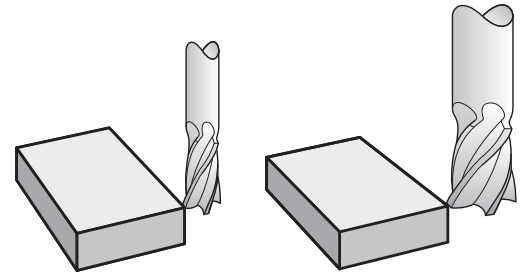


5.3 Tool compensation

5.3 Tool compensation

Introduction

The TNC 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**).

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

Compensation value = $L + DL_{\text{TOOL CALL}} + DL_{\text{TAB}}$ with

L: Tool length **L** from **TOOL DEF** block or tool table

DL_{TOOL CALL}: Oversize for length **DL** in the **TOOL CALL** block

DL_{TAB}: Oversize for length **DL** in the tool table

Tool radius compensation with paraxial positioning blocks

The TNC can compensate the tool radius in the machining 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 TNC takes the delta values from both the **TOOL CALL** block and the tool table into account:

Compensation value = **R** + **DR_{TOOL CALL}** + **DR_{TAB}** with

R: Tool radius **R** from **TOOL DEF** block or tool table

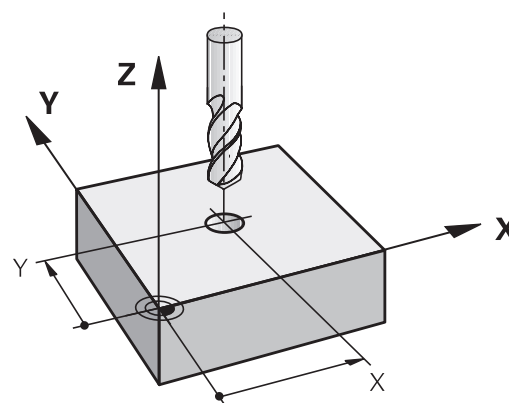
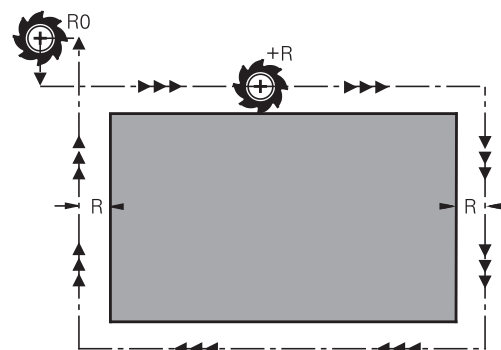
DR_{TOOL CALL}: Oversize for radius **DR** in the **TOOL CALL** block

DR_{TAB}: Oversize for radius **DR** in the tool table

Contouring without radius compensation: R0

The tool center moves on the machining plane along the to the programmed coordinates.

Applications: Drilling and boring, pre-positioning







5.3 Tool compensation

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?

- | | |
|---|---|
|  | ▶ The TNC extends the traverse path of the tool by the tool radius |
|  | ▶ The TNC shortens the traverse path of the tool by the tool radius |
|  | ▶ Select tool movement without radius compensation or cancel radius compensation:
Press the ENT key |
|  | ▶ Terminate the block: Press the END key |

6

**Programming tool -
movements**

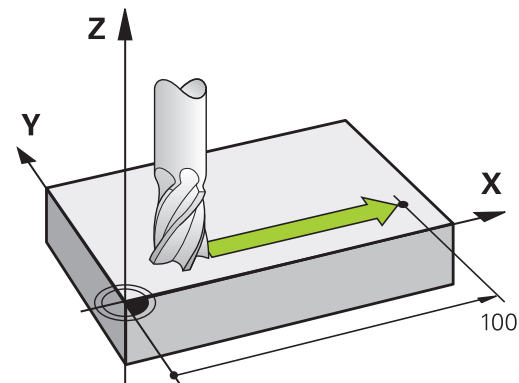
6 Programming tool movements

6.1 Fundamentals

6.1 Fundamentals

Tool movements in the program

The orange axis keys initiate the dialog for a paraxial positioning block. The TNC asks you successively for all the necessary information and inserts the NC block into the machining 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.



Danger of collision!

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

Radius compensation

The TNC can compensate the tool radius automatically. In paraxial positioning blocks you can select whether the TNC is to extend (R+) or shorten (R-) the traverse path by the value of the tool radius.

Further Information: "Tool radius compensation with paraxial positioning blocks", page 199

Miscellaneous functions M

With the TNC'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 211

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 229

6 Programming tool movements

6.2 Tool movements


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 TNC asks you successively for all the necessary information and inserts the NC block into the machining program.

Example—programming a straight line


-  ▶ 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

-  ▶ Press the **ENT** key


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


-  ▶ Select the radius compensation (here, press 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.)


-  ▶ Confirm your entry with the **ENT** key, or

-  ▶ Move at rapid traverse: Press the **F MAX** soft key, or

-  ▶ Traverse with the feed rate defined in the **TOOL CALL** block: Press the **F AUTO** soft key

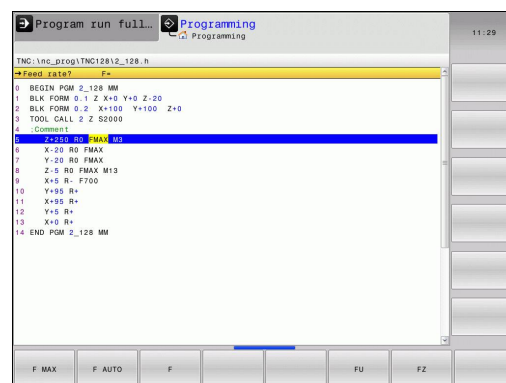
MISCELLANEOUS FUNCTION M?

- ▶ **3** (enter the miscellaneous function **M3** “spindle on”)

-  ▶ The TNC 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 program block after which you want to insert the block

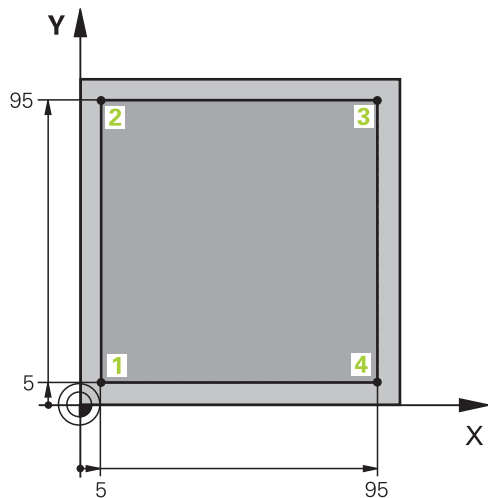


- ▶ Press the **ACTUAL-POSITION-CAPTURE** key for the TNC to generate an block
- ▶ Select the desired axis, e.g. by pressing the **ACTL. POS. X** soft key: The TNC adopts the current position and ends the dialog

6 Programming tool movements

6.2 Tool movements

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**

Data transfer from CAD files

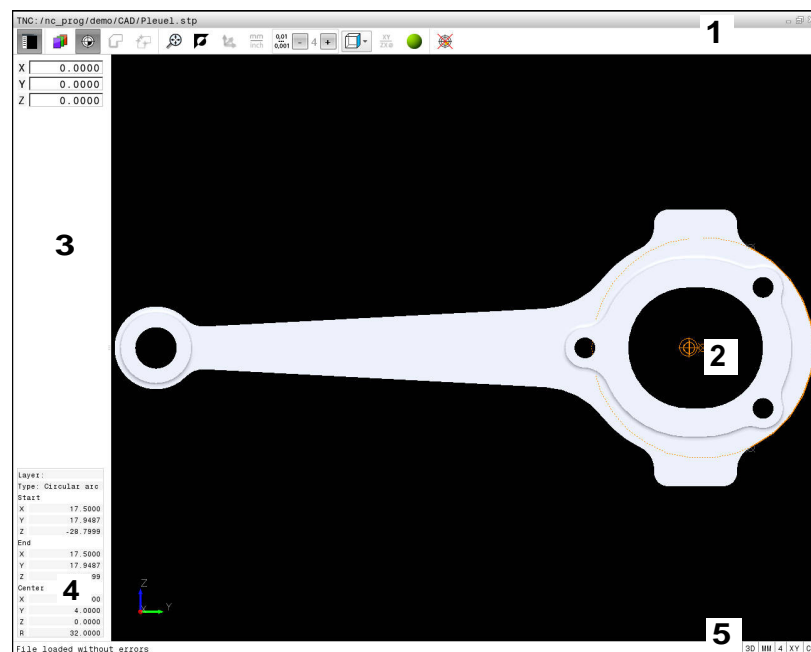
7.1 CAD viewer and

7.1 CAD viewer and

Fundamentals of the CAD viewer and

Screen display

If 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 allow you to open standardized CAD data formats directly on the TNC.

The TNC 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
		■ R12
		■ R13
		■ 2000
		■ 2002









7.2 CAD viewer

Application

The file can simply be selected via the file manager of the TNC, just like NC programs. This allows you to view models quickly and easily.

The datum can be positioned anywhere in the model. Starting from this datum, element information such as centers of circles can be shown.

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 the datum or delete set datum
	
	Set the zoom to the largest possible view of the complete graphics
	Change the background color (black or white)
	Set resolution: The resolution specifies how many decimal places the TNC 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

8

**Subprograms and
program section
repeats**

Subprograms and program section repeats

8.1 Labeling 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.



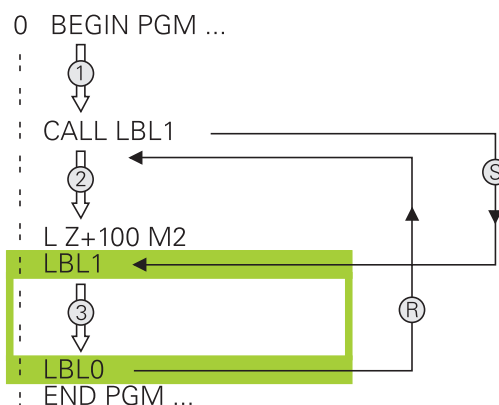
Do not use a label number or label name more than once!

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 TNC executes the part program up to calling a subprogram, **CALL LBL**.
- 2 The subprogram is then executed from beginning to end, **LBL 0**.
- 3 The TNC 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

Program the subprogram

LBL
SET

- ▶ 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**

Subprograms and program section repeats

8.2 Subprograms

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 TNC will then jump 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

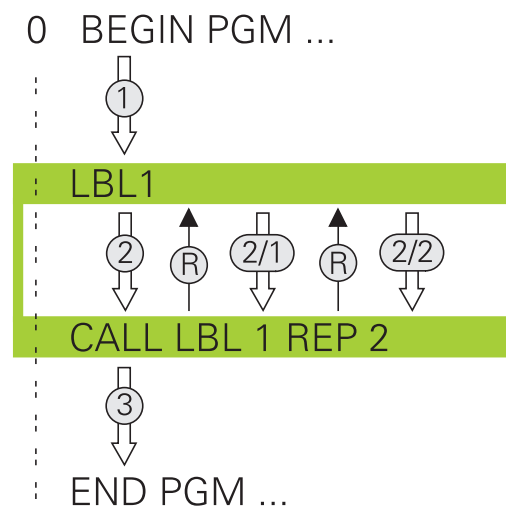


CALL LBL 0 is not permitted (Label 0 is only used to mark the end of a subprogram).

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 TNC 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 TNC resumes the part program after the last repetition

Programming notes

- You can repeat a program section up to 65 534 times in succession
- The total number of times the program section is executed is always one more than the programmed number of repeats, because the first repeat starts after the first machining process.

Subprograms and program section repeats

8.3 Program-section repeats

Programming a program section repeat

LBL
SET

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

Overview of the soft keys

If the **PGM CALL** key is pressed, the TNC displays the following soft keys:

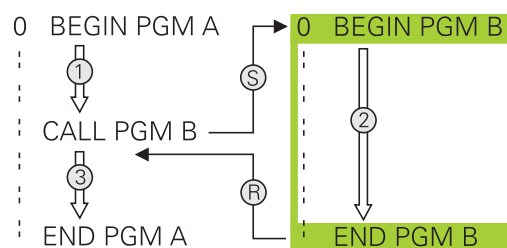
Soft key	Function
CALL PROGRAM	Call a program with PGM CALL
SELECT DATUM TABLE	Select a datum table with SEL TABLE
SELECT POINT TABLE	Select a point table with SEL PATTERN
SELECT PROGRAM	Select a program with SEL PGM
CALL SELECTED PROGRAM	Select last selected file with CALL SELECTED PGM

Subprograms and program section repeats

8.4 Any desired program as subprogram

Operating sequence

- 1 The TNC executes the part program up to the block in which another program is called with **CALL PGM**
- 2 Then the other part program is run from beginning to end
- 3 The TNC then resumes the first part program (i.e. the calling program) with the block after the program call



Programming notes

- The TNC does not need any labels to call any part program
- The called program must not contain the miscellaneous functions **M2** or **M30**. If you have defined subprograms with labels in the called part program, you then need to replace M2 or M30 with the **FN 9: IF +0 EQU +0 GOTO LBL 99** jump function to force a jump over this program section
- The called part program must not contain a **CALL PGM** call into the calling part program, otherwise an infinite loop will result

Calling any program as a subprogram



Danger of collision!

Coordinate transformations that you define in the called program also remain in effect for the calling program too, unless you reset them.



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 program you are calling it from, you must enter the complete path, for example **TNC: \ZW35\SCHRUPP\PGM1.H**

You can also call a program with Cycle **12 PGM CALL**.

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.

Call 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 TNC 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 TNC displays a selection window in which you can select the program to be called; confirm with the **ENT** key

Subprograms and program section repeats




8.4 Any desired program as subprogram

Call 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:

- 
 - ▶ To select the functions for program call, press the **PGM CALL** key
- 
 - ▶ Press the **SELECT PROGRAM** soft key: The TNC starts the dialog for defining the program to be called
- 
 - ▶ Press the **SELECT FILE** soft key: The TNC displays a selection window in which you can select the program to be called; confirm with the **ENT** key

To call the selected program, proceed as follows:

- 
 - ▶ To select the functions for program call, press the **PGM CALL** key
- 
 - ▶ Press the **CALL SELECTED PROGRAM** soft key: The TNC calls the last program selected with **CALL SELECTED PGM**

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

Subprograms and program section repeats

8.5 Nesting

Subprogram within a subprogram

Example NC blocks

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 NC blocks

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.

Subprograms and program section repeats

8.5 Nesting

Repeating a subprogram

Example NC blocks

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

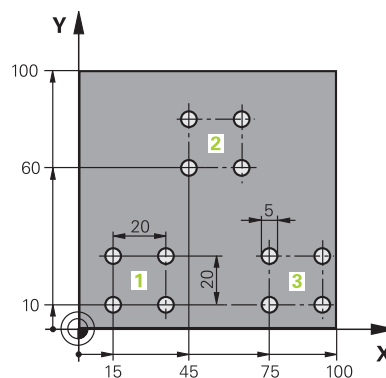
- 1 Main program UPGREP is executed up to block 11.
- 2 Subprogram 2 is called and executed.
- 3 Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.
- 4 Main program UPGREP is executed from block 13 up to block 19. Return jump to block 1 and 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	

Subprograms and program section repeats

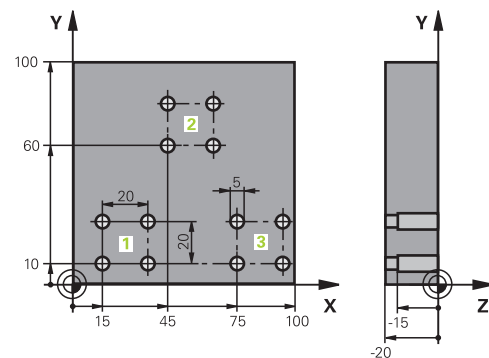
8.6 Programming examples

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

Subprograms and program section repeats

8.6 Programming examples

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**

Programming Q parameters

9.1 Principle and overview of functions

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.:

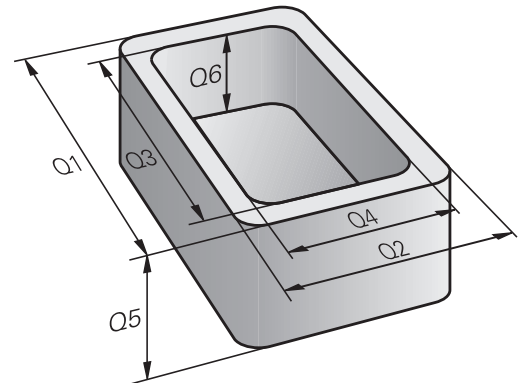
- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

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 effect all NC programs in the TNC memory
	0 – 99	Parameters for the user , if there are no overlaps with the HEIDENHAIN-SL cycles
	100 – 199	Parameters for system information on the TNC that can be read by the 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 that are nonvolatile on all NC programs in the TNC memory, i.e. they remain in effect even after a power interruption
	0 – 499	Parameters for users

QS parameters (the **S** stands for string) are also available on the TNC and enable you to process texts.

Q parameter type	Q parameter range	Meaning
QS parameters:		Parameters effect all NC programs in the TNC memory
	0 – 99	Parameters for the user , where no overlaps with the HEIDENHAIN SL cycles are present
	100 – 199	Parameters for diverse functions on the TNC that can be read by the 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



You gain maximum safety for your applications by using only Q parameter ranges recommended for the user in your NC programs.

Please note that the specified use of the Q parameter ranges is recommended by HEIDENHAIN but cannot be ensured.

Machine manufacturer or third-party functions may still cause overlaps with the user's NC program. Please refer to the machine manual and third-party documentation for this.

Programming Q parameters

9.1 Principle and overview of functions

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 TNC calculates numbers up to a value of 10^{10} .

You can assign a maximum of 255 characters to **QS**QS parameters.



The TNC automatically assigns some Q and QS parameters the same data, e.g. the Q parameter **Q108** is automatically assigned the current tool radius.


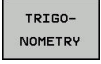
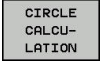

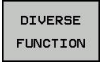

Further Information: " Preassigned Q parameters", page 291

The TNC saves numerical values internally in a binary number format (standard IEEE 754). Due to this standardized format some decimal numbers do not have an exact binary representation (round-off error). Keep this in mind especially when you use calculated Q-parameter contents for jump commands or positioning movements.

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 TNC then displays the following soft keys:

Soft key	Function group	Page
	Basic arithmetic (assign, add, subtract, multiply, divide, square root)	235
	Trigonometric functions	238
	Function for calculating circles	239
	If/then conditions, jumps	240
	Other functions	244
	Entering formulas directly	275



The TNC shows the soft keys Q, QL and QR when you are defining or assigning a Q parameter. First press one of these soft keys to select the desired type of parameter, and then enter the parameter number.

If you have a USB keyboard connected, you can press the **Q** key to open the dialog for entering a formula.

Programming Q parameters

9.2 Part families—Q parameters in place of numerical values

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 NC blocks

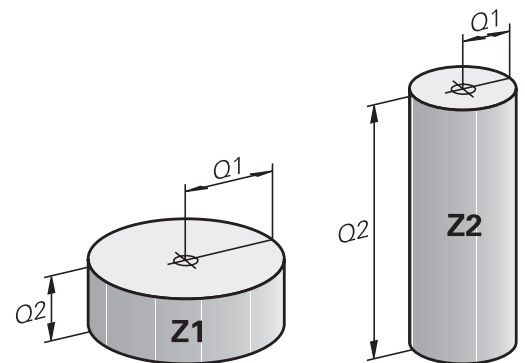
15 FN 0: Q10=25	Assign
...	Q10 is assigned the value 25
25 X +Q10	Means X +25

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 mathematical functions, press the **BASIC ARITHM.** soft key. The TNC then displays the following soft keys:

Overview

Soft key	Function
<div>FN0</div> <div>X = Y</div>	FN 0: ASSIGN e. g. FN 0: Q5 = +60 Directly assign value Reset Q parameter value
<div>FN1</div> <div>X + Y</div>	FN 1: ADDITION e.g. FN 1: Q1 = -Q2 + -5 Form and assign sum from two values
<div>FN2</div> <div>X - Y</div>	FN 2: SUBTRACTION e. g. FN 2: Q1 = +10 - +5 Form and assign difference between two values
<div>FN3</div> <div>X * Y</div>	FN 3: MULTIPLICATION e. g. FN 3: Q2 = +3 * +3 Form and assign the product of two values
<div>FN4</div> <div>X / Y</div>	FN 4: DIVISION e.g. FN 4: Q4 = +8 DIV +Q2 Form and assign the quotient of two values Not permitted: Division by 0
<div>FN5</div> <div>SQRT</div>	FN 5: SQUARE ROOT e.g. FN 5: Q20 = SQRT 4 Form and assign the square root of a value Not permitted: Square root from negative value

To the right of the "=" character you can enter the following:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming Q parameters

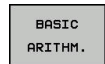
9.3 Describing contours with mathematical functions

Programming fundamental operations

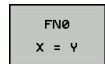
Example 1



- ▶ Select the Q parameter function: Press the **Q** key



- ▶ To select the mathematical functions, press the **BASIC ARITHM.** soft key.



- ▶ Select the ASSIGN Q parameter function: Press the **FN0 X = Y** soft key

NC sets in the TNC

16 FN 0: Q5 = +10

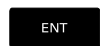
17 FN 3: Q12 = +Q5 * +7

PARAMETER NUMBER FOR RESULT?



- ▶ Enter **12** (the number of the Q parameter) and confirm with the **ENT** key

FIRST VALUE / PARAMETER?

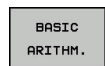


- ▶ Enter **10**: Assign the numerical value 10 to Q5 and confirm with the **ENT** soft key

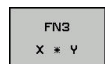
Example 2



- ▶ Select the Q parameter function: Press the **Q** key



- ▶ To select the mathematical functions, press the **BASIC ARITHM.** soft key.



- ▶ To select the MULTIPLICATION Q parameter function, press the **FN3 X * Y** soft key

PARAMETER NUMBER FOR RESULT?



- ▶ Enter **12** (the number of the Q parameter) and confirm with the **ENT** key

FIRST VALUE / PARAMETER?



- ▶ Enter **Q5** as the first value and confirm with the **ENT** key.

SECOND VALUE / PARAMETER?



- ▶ Enter **7** as the second value and confirm with the **ENT** key.

Describing contours with mathematical functions 9.3

Example 3 – Reset Q parameters

- ▶ Select the Q parameter function: Press the **Q** key
- ▶ To select the mathematical functions, press the **BASIC ARITHM.** soft key.
- ▶ To select the ASSIGN Q parameter function, press the **FN0 X = Y** soft key

NC sets in the TNC

16 FN 0: Q5 SET UNDEFINED

16 FN 0: Q1 = Q5

PARAMETER NUMBER FOR RESULT?

- ▶ Enter **12** (the number of the Q parameter) and confirm with the **ENT** key

1. VALUE OR PARAMETER?

- ▶ 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

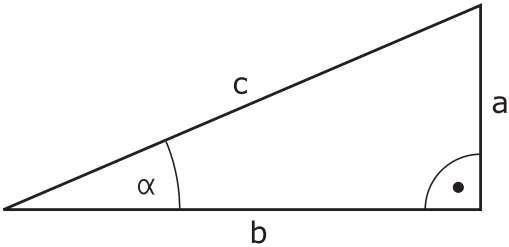
- Sine:** $\sin \alpha = a / c$
- Cosine:** $\cos \alpha = b / c$
- Tangent:** $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

The TNC can find the angle from the tangent:

$\alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$



Example:

a = 25 mm
b = 50 mm
 $\alpha = \arctan (a / b) = \arctan 0.5 = 26.57^\circ$
Furthermore:
 $a^2 + b^2 = c^2$ (where $a^2 = a \times a$)
 $c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

Press the **TRIGONOMETRY** soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table below.

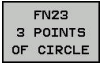
Soft key	Function
<div>FN6 SIN(X)</div>	FN 6: SINUS e. g. FN 6: Q20 = SIN-Q5 Define and assign the sine of an angle in degrees (°)
<div>D7 COS(X)</div>	FN 7: COSINUS e. g. FN 7: Q21 = COS-Q5 Define and assign the cosine of an angle in degrees (°)
<div>FN8 X LEN Y</div>	FN 8: ROOT SUM OF SQUARES e. g. FN 8: Q10 = +5 LEN +4 Form and assign length from two values
<div>FN13 X ANG Y</div>	FN 13: ANGLE e. g. FN 13: Q20 = +25 ANG-Q1 Calculate the angle from the arc tangent of the opposite and adjacent sides or from the sine and cosine of the angle (0 < angle < 360°) and assign it to a parameter

9.5 Calculation of circles

Application

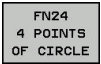
The TNC can use the functions for calculating circles to calculate the circle center and the circle radius from three or four given points on the circle. The calculation is more accurate if four points are used.

Application: These functions can be used if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Soft key	Function
	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 TNC 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
	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 TNC 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.

Programming Q parameters

9.6 If-then decisions with Q parameters

9.6 If-then decisions with Q parameters

Application

The TNC can make logical if-then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition.

Further Information: "Labeling subprograms and program section repeats", page 212

If it is not fulfilled, the TNC continues with 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 TNC then displays the following soft keys:

Soft key	Function
<div>FN9 IF X EQ Y GOTO</div>	FN 9: IF EQUAL, JUMP e. g. FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"
<div>EQU</div>	If both values or parameters are equal, jump to specified label
<div>FN9 IF X EQ Y GOTO</div>	FN 9: IF UNDEFINED, JUMP e. g. FN 9: IF +Q1 IS UNDEFINED GOTO LBL "UPCAN25"
<div>IS UNDEFINED</div>	If the given parameter is undefined, jump to the specified label
<div>FN9 IF X EQ Y GOTO</div>	FN 9: IF DEFINED, JUMP e. g. FN 9: IF +Q1 IS DEFINED GOTO LBL "UPCAN25"
<div>IS DEFINED</div>	If the given parameter is defined, jump to the specified label
<div>FN10 IF X NE Y GOTO</div>	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
<div>FN11 IF X GT Y GOTO</div>	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
<div>FN12 IF X LT Y GOTO</div>	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

Programming Q parameters

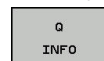
9.7 Checking and changing Q parameters

9.7 Checking and changing Q parameters

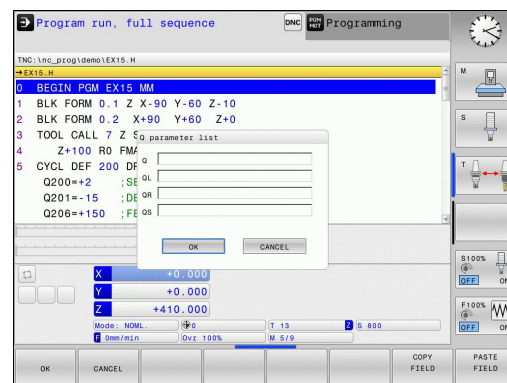
Procedure

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 STOP** 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 TNC lists all parameters and their 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



The parameters used by the TNC internally or in cycles are provided with comments.

If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The TNC 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).

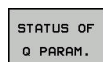
- ▶ If you are in a program run, interrupt it if required (e.g. by pressing the **NC STOP** key and the **INTERNAL STOP** soft key), or stop the test run



- ▶ Call the soft key row for screen layout



- ▶ Select the screen layout with additional status display: In the right half of the screen, the TNC shows the **Overview** status form



- ▶ Press the **STATUS OF Q PARAM.** soft key **STATUS OF Q PARAM.**



- ▶ Press the **Q PARAMETER LIST** soft key: The TNC 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} .

Programming Q parameters

9.8 Additional functions

9.8 Additional functions

Overview

Press the **DIVERSE FUNCTION** soft key to call the additional functions. The TNC then displays the following soft keys:

Soft key	Function	Page
FN14 ERROR=	FN 14: ERROR Display error messages	245
FN16 F-PRINT	FN 16: F-PRINT Formatted output of texts or Q parameter values	249
FN18 SYS-DATUM READ	FN 18: SYSREAD Read system data	254
FN19 PLC=	FN 19: PLC Transfer values to the PLC	262
FN20 WAIT FOR	FN 20: WAIT FOR NC and PLC synchronization	262
FN26 OPEN TABLE	FN 26: TABOPEN Open a freely definable table	312
FN27 WRITE TO TABLE	FN 27: TABWRITE Write to a freely definable table	313
FN28 READ FROM TABLE	FN 28: TABREAD Read from a freely definable table	314
FN29 PLC LIST=	FN 29: PLC Transfer up to eight values to the PLC	263
FN37 EXPORT	FN 37: EXPORT Export local Q parameters or QS parameters into a calling program	263
FN38 SEND	FN 38: SEND Send information from the NC program	263

FN 14: ERROR: Displaying error messages

With the **FN 14: ERROR** function you can call messages under program control. The messages are predefined by the machine manufacturer or by HEIDENHAIN. Whenever the TNC comes to a block with **FN 14: ERROR** in the Program Run or Test Run mode, it interrupts the program run and displays a message. The program must then be restarted.

Error numbers area	Standard dialog
0 ... 999	Machine-dependent dialog
1000 ... 1199	Internal error messages

Example NC block

The TNC is to display the message stored under error number 1000.

```
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

9.8 Additional functions

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

9.8 Additional functions

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 text and Q parameter values



With **FN16: F-PRINT**, you can also output to the screen any messages from the NC program. Such messages are displayed by the TNC in a pop-up window.

The function **FN16: F-PRINT** transfers Q parameter values and texts in a selectable format. If you send the values, the TNC saves the data in the file that you defined in the **FN16** block. The maximum size of the output file is 20 kilobytes.

To be able to use the function **FN16: F-PRINT**, first program a text file that defines the output format.

Available functions

When you create a text file, use the following formatting functions:

Special characters	Function
"....."	Define output format for texts and variables between the quotation marks
%9.3F	Format for Q parameter: <ul style="list-style-type: none"> ■ Define %: format ■ 9.3: 9 total characters (incl. decimal point), of which 3 are places after the decimal point ■ F: Floating (decimal number), format for Q, QL, QR
%+7.3F	Format for Q parameter: <ul style="list-style-type: none"> ■ Define %: format ■ +: number right-aligned ■ 7.3: 7 total characters (incl. decimal point), of which 3 are places after the decimal point ■ 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

9.8 Additional functions

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 Rumanian conversational language

Keyword	Function
L_SLOVAK	Outputs text only for Slovak 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 TNC'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";

"DATUM: %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;

Programming Q parameters

9.8 Additional functions

In the part program, program **FN 16: F-PRINT** to activate the output:

```
96 FN 16: F-PRINT TNC:\MASK\MASK1.A/ TNC:\PROT1.TXT
```

The TNC 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



If you output the same file more than once in the program, the TNC appends all texts to the end of the texts already output within the target file.

If you use **FN16** more than once in the program, the TNC saves all texts in the file that you defined in the **FN16** function. The file is not output until the TNC reads the **END PGM** block, or you press the **NC STOP** key, or you close the file with **M_CLOSE**.

In the **FN16** block, program the format file and the log file with their respective file type extensions

If you enter only the file name for the path of the log file, the TNC saves the log file in the directory in which the NC program with the **FN16** function is located.

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.

Displaying messages on the TNC 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 TNC screen. This makes it easy to display explanatory texts, including long texts, at any point in the program in a way that the operator 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 TNC screen, you need only enter **SCREEN:** as the name of the protocol file.

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:

96 FN 16: F-PRINT TNC:\MASK\MASK1.A/SCLR:



As standard behavior, the **FN16** function overwrites already existing log files with the same name Use **M_APPEND** if you wish to append logs to existing logs upon renewed output.

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:

96 FN 16: F-PRINT TNC:\MSK\MSK1.A / PC325:\LOG\PRO1.TXT



As standard behavior, the **FN16** function overwrites already existing log files with the same name Use **M_APPEND** if you wish to append logs to existing logs upon renewed output.

9.8 Additional functions

FN 18: SYSREAD – Reading system data

With the **FN 18: SYSREAD** function you can read system data and store them in Q parameters. You select the system data through a group name (ID number), and additionally through a number and an index.



Values read from the function **FN 18: SYSREAD** are always output in metric units.

Group name, ID no.	Number	Index	Meaning
Program information, 10	3	-	Number of the active fixed cycle
	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.
System jump addresses, 13	1	-	Label jumped to during M2/M30 instead of ending the current program Value = 0: M2/M30 functions normally
	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) instead of aborting the program with an error message. Value = 0: Server error has the normal effect.
Machine status, 20	1	-	Active tool number (without index)
	2	-	Prepared tool number (without index)
	3	-	Active tool axis 0=X, 1=Y, 2=Z, 6=U, 7=V, 8=W
	4	-	Programmed spindle speed
	5	-	Active spindle condition: -1=not defined, 0=M3 active, 1=M4 active, 2=M5 after M3, 3=M5 after M4
	7	-	Gear range
	8	-	Coolant status: 0=off, 1=on
	9	-	Active feed rate
	10	-	Index of prepared tool
	11	-	Index of active tool
Channel data, 25	1	-	Channel number
Cycle parameter, 30	1	-	Set-up clearance of active fixed cycle
	2	-	Drilling depth or milling depth in active fixed cycle

Group name, ID no.	Number	Index	Meaning
	3	-	Plunging depth of active machining cycle
	4	-	Feed rate for pecking in active fixed cycle
	5	-	1st side length for rectangular pocket cycle
	6	-	2nd side length for rectangular pocket cycle
	7	-	1st side length for slot cycle
	8	-	2nd side length for slot cycle
	10	-	Feed rate for milling in active fixed cycle
	11	-	Direction of rotation for active fixed cycle
	12	-	Dwell time for active fixed cycle
	13	-	Thread pitch for Cycles 17
	14	-	Finishing allowance for active fixed cycle
	22	-	Probing path
	23	-	Probing feed rate
Modal condition, 35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
Data for SQL tables, 40	1	-	Result code for the last SQL command
Data from the tool table, 50	1	Tool no.	Tool length
	2	Tool no.	Tool radius
	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 (0 or 1)
	8	Tool no.	Number of the replacement tool
	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
	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: Rotational direction DIRECT (0=positive/ 1=negative)
	19	Tool no.	TT: Offset in plane R-OFFS
	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
	23	Tool no.	PLC value
	28	Tool no.	Maximum rpm NMAX

9 Programming Q parameters

9.8 Additional functions

Group name, ID no.	Number	Index	Meaning
	32	Tool no.	Point angle TANGLE
	35	Tool no.	Wear tolerance for radius R2TOL
	37	Tool no.	Corresponding line in the touch-probe table
	38	Tool no.	Timestamp of last use
Pocket table data, 51	1	Pocket number	Tool number
	2	Pocket number	Special tool: 0=No, 1=Yes
	3	Pocket number	Fixed pocket: 0=No, 1=Yes
	4	Pocket number	Locked pocket: 0=No, 1=Yes
	5	Pocket number	PLC status
Tool location, 52	1	Tool no.	Pocket number P
	2	Tool no.	Magazine number
File information, 56	1	-	Number of lines of the selected tool table
	2	-	Number of lines of the selected datum table
	4	-	Number of lines in the open, freely definable table Value -1: No table open
Values programmed immediately after tool call, 60	1	-	Tool number T
	2	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 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
	8	-	Tool index
	9	-	Active feed rate
Values programmed immediately after tool definition TOOL DEF, 61	1	-	Tool number T
	2	-	Length
	3	-	Radius
	4	-	Index
	5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name, ID no.	Number	Index	Meaning
Active 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
	3	1 = without oversize 2 = with oversize 3 = with oversize and Oversize from TOOL CALL	Rounding radius R2
Active transformations, 210	3	-	Active mirrored axis
			0: Mirroring not active
			+1: X axis mirrored
			+2: Y axis mirrored
			+4: Z axis mirrored
			+64: U axis mirrored
			+128: V axis mirrored
			+256: W axis mirrored
	Combinations = Sum of individual axes		
	4	1	Active scaling factor in X axis
4	2	Active scaling factor in Y axis	
4	3	Active scaling factor in Z axis	
4	7	Active scaling factor in U axis	
4	8	Active scaling factor in V axis	
4	9	Active scaling factor in W axis	
Active datum shift, 220	2	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis

9

Programming Q parameters

9.8 Additional functions

Group name, ID no.	Number	Index	Meaning
		7	U axis
		8	V axis
		9	W axis
	3	1 to 9	Difference between reference point and datum in axes 1 to 9
Traverse range, 230	2	1 to 9	Negative software limit switch or traverse range limit in axes 1 to 9
	3	1 to 9	Positive software limit switch or traverse range limit in axes 1 to 9
	5	-	Software limit switch on or off: 0 = on, 1 = off
Nominal position in the machine coordinate system, 240	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Current position in the active coordinate system, 270	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Machining time, 320	3	-	Momentary machining time of the active NC program in minutes
TS triggering touch probe, 350	50	1	Type of touch probe
		2	Line in the touch-probe table
	51	-	Effective length
	52	1	Effective ball radius
		2	Rounding radius
	53	1	Center offset (reference axis)
		2	Center offset (minor axis)

Group name, ID no.	Number	Index	Meaning
	54	-	Spindle-orientation angle in degrees (center offset)
	55	1	Rapid traverse
		2	Measuring feed rate
	56	1	Maximum measuring range
		2	Safety clearance
	57	1	Spindle orientation possible: 0=No, 1=Yes
		2	Spindle-orientation angle
TT tool touch probe	70	1	Type of touch probe
		2	Line in the touch-probe table
	71	1	Center point in reference axis (REF system)
		2	Center point in minor axis (REF system)
		3	Center point in tool axis (REF system)
	72	-	Plate radius
	75	1	Rapid traverse
		2	Measuring feed rate for stationary spindle
		3	Measuring feed rate for rotating spindle
	76	1	Maximum measuring range
		2	Safety clearance for linear measurement
		3	Safety clearance for radial measurement
	77	-	Spindle speed
	78	-	Probing direction
Reference point from touch probe cycle, 360	1	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length compensation but with probe radius compensation (workpiece coordinate system)
	2	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or probe radius compensation (machine coordinate system)
	3	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Result of measurement of the touch probe cycles 0 and 1 without probe radius or probe length compensation
	4	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or probe radius compensation (workpiece coordinate system)
	10	-	Oriented spindle stop
	11	-	Error status for suppressed error message 0 = Probe process successful -1 = Touch point not reached

9 Programming Q parameters

9.8 Additional functions

Group name, ID no.	Number	Index	Meaning
Value from the active datum table in the active coordinate system, 500	Line	Column	Read values
Basic transformation, 507	Line	1 to 6 (X, Y, Z, SPA, SPB, SPC)	Read the basic transformation of a preset
Axis offset, 508	Line	1 to 9 (X_OFFS, Y_OFFS, Z_OFFS, A_OFFS, B_OFFS, C_OFFS, U_OFFS, V_OFFS, W_OFFS)	Read the axis offset of a preset
Active preset, 530	1	-	Read the number of the active preset
SIK, 630	2	-	Read SIK ID
Read data of the 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
	11	-	Current tool age CUR. TIME
	12	-	PLC status
	13	-	Maximum tooth length LCUTS
	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
	20	-	TT: Offset in length L-OFFS
	21	-	TT: Breakage tolerance for length, LBREAK
	22	-	TT: Breakage tolerance for radius, RBREAK
	23	-	PLC value
	24	-	Tool type TYP 0 = Milling cutter, 21 = Touch probe

Group name, ID no.	Number	Index	Meaning
	27	-	Corresponding line in the touch-probe table
	32	-	Tip angle
Tool usage test, 975	1	-	Tool usage test of the current NC program -2= no test possible, deactivated by the machine manufacturer -1 = no test possible, no tool usage file 0 = test OK, all tools available 1 = test not OK, no tool or tool is locked
Touch probe cycles, 990	1	-	Approach behaviour: 0 = Standard behavior 1 = Effective radius, Safety clearance zero
	2	-	0 = Pushbutton monitoring off 1 = Pushbutton monitoring on
	4	-	0 = Stylus not deflected 1 = Stylus deflected
	8	-	Current spindle angle
Tool number, 990	10	Q parameter-number	Tool number associated with the tool name of the Q parameter IDX -1 = name is not available or tool is locked
Execution status, 992	10	-	Mid-program startup active 1 = Yes, 0 = No
	11	-	Search phase
	14	-	Number of the last FN14 error
	16	-	Real execution active 1 = execution , 0 = simulation
	31	-	Radius compensation in MDI mode with paraxial positioning blocks permitted 0 = Not permitted, 1 = Permitted

Example: Assign the value of the active scaling factor for the Z axis to Q25.

55 FN 18: SYSREAD Q25 = ID210 NR4 IDX3

Programming Q parameters

9.8 Additional functions

FN 19: PLC – Transfer values to the PLC



This function may only be used with the permission of your machine tool builder.

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



This function may only be used with the permission of your machine tool builder.

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 **FN18: SYSREAD** that require synchronization with real time. The TNC stops the look-ahead calculation and executes the subsequent 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



This function may only be used with the permission of your machine tool builder.

The **FN 29: PLC** function transfers up to eight numerical values or Q parameters to the PLC.

FN 37: EXPORT



This function may only be used with the permission of your machine tool builder.

You need the **FN 37: EXPORT** function if you want to create your own cycles and integrate them in the TNC.

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
```

Programming Q parameters

9.9 Accessing tables with SQL commands

9.9 Accessing tables with SQL commands

Introduction

Accessing of tables is programmed on the TNC with SQL commands in **transactions**. A transaction consists of multiple SQL commands that guarantee an orderly execution of the table entries.



Tables are configured by the machine manufacturer. Names and designations required as parameters for SQL commands are also specified.

The following **terms** are used:

- **Table:** A table consists of x columns and y rows. It is saved as a file in the File Manager of the TNC, and is addressed with the path and file name (= table name). Synonyms can also be used for addressing, as an alternative to the path and file name
- **Columns:** The number and names of the columns are specified when configuring the table. The column name is used for addressing in various SQL commands
- **Rows:** The number of rows is variable. You can insert new rows. There are no row numbers or other designators. You can however select rows according to their column content. Rows can only be deleted in the table editor and not via the NC program.
- **Cell:** The column in a row
- **Table entry:** Content of a cell
- **Result set:** During a transaction, the selected columns and rows are managed in the result set. You can view the result set as a sort of "intermediate memory," which temporarily assumes the set of selected columns and rows. Result set
- **Synonym:** This term defines a name used for a table instead of its path and file name. Synonyms are specified by the machine manufacturer in the configuration data

A transaction

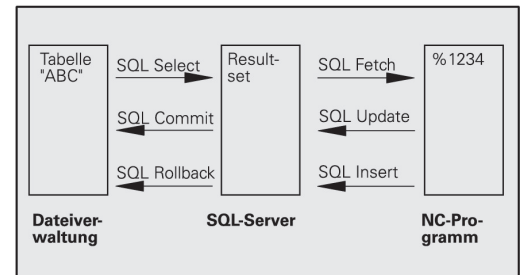
In principle, a transaction consists of the following actions:

- Address the table (file), select rows and transfer them to the result set
- Read rows from the result set, change rows or insert new rows
- Conclude transaction. If changes/insertions were made, the rows from the result set are transferred to the table (file)

Other actions are also necessary so that table entries can be edited in an NC program and to ensure that other changes are not made to copies of the same table rows at the same time. This results in the following **transaction sequence**:

- 1 A Q parameter is specified for each column to be edited. The Q parameter is assigned to a column—it is "bound" (**SQL BIND...**)
- 2 Address the table (file), select rows and transfer them to the result set. In addition, you define which columns are transferred to the result set (**SQL SELECT...**). You can lock the selected rows. Other processes can then read these rows, but cannot change the table entries. You should always lock the selected rows when you are going to make changes (**SQL SELECT ... FOR UPDATE**)
- 3 Read, change and/or add new lines in the results set: – Copy a line from the results set into the Q parameter of your NC program (**SQL FETCH...**) – Prepare changes in the Q parameters and transfer one row from the result set (**SQL UPDATE...**) – Prepare new table row in the Q parameters and transfer into the result set as a new row (**SQL INSERT...**)
- 4 Conclude transaction. – If changes/insertions have been made, the data from the result set is placed in the table (file). The data is now saved in the file. Any locks are canceled, and the result set is released (**SQL COMMIT...**). – If table entries have **not** been changed or inserted (read-only access), any locks are canceled and the result set is released (**SQL ROLLBACK... WITHOUT INDEX**)

Multiple transactions can be edited at the same time.



You must conclude a transaction, even if it consists solely of read accesses. This is to ensure that modifications/enhancements are not lost, locks not canceled or the result set enabled.

Programming Q parameters

9.9 Accessing tables with SQL commands

Result set

The selected rows are numbered in ascending order within the result set, starting from 0. This numbering is referred to as the **index**. The index is used for read and write accesses, enabling a row of the result set to be specifically addressed.

It can often be advantageous to sort the rows in the result set. Do this by specifying the table column containing the sorting criteria. Also select ascending or descending order (**SQL SELECT ... ORDER BY ...**).

The selected rows that were transferred to the result set are addressed with the **HANDLE**. All following SQL commands use the handle to refer to this "set of selected columns and rows."

When concluding a transaction, the handle is released (**SQL COMMIT...** or **SQL ROLLBACK...**). It is then no longer valid.

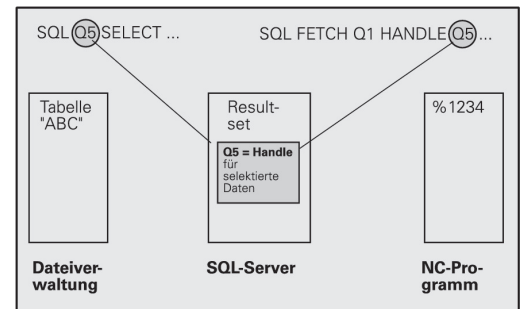
You can edit more than one result set at the same time. The SQL server assigns a new handle for each "Select" command.

"Binding" Q parameters to columns

The NC program does not have direct access to the table entries in the result set. The data must be transferred in Q parameters. In the other direction, the data is first prepared in the Q parameters and then transferred to the result set.

Specify with **SQL BIND ...** which table columns are mapped to which Q parameters. The Q parameters are "bound" (assigned) to the columns. Columns that are not bound to Q parameters are not included in the read-/write-processes.

If a new table row is generated with **SQL INSERT...**, the columns not linked to Q parameters are filled with default values.



Programming SQL commands



This function can only be programmed if you have entered the code number 555343.

Program SQL commands in the **Programming** mode:

SPEC
FCT

- ▶ Press the **SPEC FCT** key

PROGRAM
FUNCTIONS

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



- ▶ Shift the soft-key row

SQL

- ▶ To select the MOD functions, press **SQL**
- ▶ Select an SQL command using a soft key or press the **SQL EXECUTE** soft key and program the SQL command



If a table is read from or written to with SQL commands, metric units are always output.

Programming Q parameters

9.9 Accessing tables with SQL commands

Overview of the soft keys

Soft key	Function
SQL BIND	SQL BIND Bind a Q parameter to a table column
SQL SELECT	SQL SELECT Select table rows
SQL EXECUTE	SQL EXECUTE Program a Select command
SQL FETCH	SQL FETCH Read table rows from the result set and save them in Q parameters
SQL ROLLBACK	SQL ROLLBACK <ul style="list-style-type: none"> ■ If INDEX is not programmed: Discard any changes/insertions and conclude the transaction ■ If INDEX is programmed: The indexed row remains in the result set – all other rows are deleted from the result set The transaction is not concluded
SQL COMMIT	SQL COMMIT Transfer table rows from the result set into the table and conclude the transaction.
SQL UPDATE	SQL UPDATE Save data from the Q parameters in an existing table row in the result set
SQL INSERT	SQL INSERT Save data from the Q parameters in a new table row in the result set

SQL BIND

SQL BIND links a Q parameter to a table column. The SQL commands "Fetch," "Update" and "Insert" evaluate this binding (assignment) during data transfer between the result set 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.



- You can program any number of bindings. Read and write processes only take into account the columns that were entered in the "Select" command.
- **SQL BIND...** must be programmed **before** "Fetch," "Update" or "Insert" commands are programmed. You can program a "Select" command without a preceding "Bind" command.
- If in the "Select" command you include columns for which no link has been programmed, an error occurs during read/write processes (program interrupt).

SQL
BIND

- ▶ **Parameter no. for result:** Q parameter that is linked (assigned) to the table column
- ▶ **Database: column name:** Enter the table name and column description – separated by . a
table name: synonym or path and file name of this table. The synonym is entered directly – path and file name are entered in single speech marks
Column description: description of the table column set in the configuration data

Link Q parameters to table columns

11SQL BIND Q881
"TAB_EXAMPLE.MEAS_NO"

12SQL BIND Q882
"TAB_EXAMPLE.MEAS_X"

13SQL BIND Q883
"TAB_EXAMPLE.MEAS_Y"

14SQL BIND Q884
"TAB_EXAMPLE.MEAS_Z"

Cancel binding

91 SQL BIND Q881

92 SQL BIND Q882

93 SQL BIND Q883

94 SQL BIND Q884

Programming Q parameters

9.9 Accessing tables with SQL commands

SQL SELECT

SQL SELECT selects table rows and transfers them to the result set.

The SQL server places the data in the result set row-by-row. The rows are numbered in ascending order, starting from 0. This row number, called the **INDEX**, is used in the SQL commands "Fetch" and "Update".

Enter the selection criteria in the **SQL SELECT...WHERE...** function. This lets you restrict the number of rows to be transferred. If you do not use this option all rows in the table are loaded.

Enter the sorting criteria in the **SQL SELECT...ORDER BY...** function. This consists of the column designation and the keyword for ascending/descending order. If you do not use this option, the rows are placed in random order.

Lock out the selected rows for other applications with the **SQL SELECT...FOR UPDATE** function. Other applications can continue to read these rows, but cannot change them. We strongly recommend using this option if you are making changes to the table entries.

Empty result set: If no rows match the selection criteria, the SQL server returns a valid handle but no table entries.



- ▶ **Parameter no. for result:** Q parameter for the handle. The SQL server returns the handle for the group of columns and rows selected with the current "Select" command.
In the event of an error (selection could not be executed) the SQL server returns a 1. Code 0 identifies an invalid handle.
- ▶ **Data bank: SQL command text:** with the following elements:
 - **SELECT** (keyword):
Name of the SQL command, names of the table columns to be transferred. Separate column names with a , (comma). Q parameters must be linked to all columns entered here
 - **FROM** table name:
Synonym or path and file name of this table. The synonym is entered directly: The path name and table name are entered in single quotation marks in the SQL command; names of the table columns to be transferred – separate several columns with a comma. Q parameters must be bound to all columns entered here.

Select all table rows

```
11SQL BIND Q881
   "TAB_EXAMPLE.MEAS_NO"
12SQL BIND Q882
   "TAB_EXAMPLE.MEAS_X"
13SQL BIND Q883
   "TAB_EXAMPLE.MEAS_Y"
14SQL BIND Q884
   "TAB_EXAMPLE.MEAS_Z"
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE"
```

Selection of table rows with the WHERE function

```
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE
   WHERE MEAS_NO<20"
```

Selection of table rows with the WHERE function and Q parameters

```
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE
   WHERE MEAS_NO==:'Q11'"
```

Table name defined with path and file name

```
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM 'V:\TABLE
   \TAB_EXAMPLE' WHERE
   MEAS_NO<20"
```

- Optional:
WHERE selection criteria: A selection criterion consists of a column name, condition and comparator. Link several selection criteria with logical AND or OR. Program the comparative value directly or with a Q parameter. A Q parameter is introduced with a colon and placed in single quotation marks
- Optional:
ORDER BY column name **ASC** for ascending sorting, or **ORDER BY** column name **DESC** for descending sorting. If you program neither **ASC** nor **DESC**, ascending sorting is executed by default. The TNC places the selected rows in the indicated column.
- Optional:
FOR UPDATE (keyword): The selected rows are locked against write-accesses from other processes.

Condition	Programming
Equal to	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

Programming Q parameters

9.9 Accessing tables with SQL commands

SQL FETCH

SQL FETCH reads the row addressed with **INDEX** from the result set, and places the table entries in the linked (assigned) Q parameters. The result set is addressed with the **HANDLE**.
SQL FETCH takes into account all columns entered in the "Select" command.



- ▶ **Parameter no. for result:** Q parameter in which the SQL server reports the result:
0: No error
1: Error occurred (incorrect handle or index too large)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set
Further Information: "SQL SELECT", page 270
- ▶ **Database: Index for SQL result:** Row number within the result set. The table entries of this row are read and are transferred into the bound Q parameters. If you do not specify the index, the first row (n=0) is read.
Either enter the row number directly or program the Q parameter containing the index

Row number is transferred to a Q parameter

```
11SQL BIND Q881
   "TAB_EXAMPLE.MEAS_NO"
12SQL BIND Q882
   "TAB_EXAMPLE.MEAS_X"
13SQL BIND Q883
   "TAB_EXAMPLE.MEAS_Y"
14SQL BIND Q884
   "TAB_EXAMPLE.MEAS_Z"
...
20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE"
...
30 SQL FETCH Q1HANDLE Q5 INDEX
   +Q2
```

Row number is programmed directly

```
...
30 SQL FETCH Q1HANDLE Q5 INDEX5
```

SQL UPDATE

SQL UPDATE transfers the data prepared in the Q parameters to the row of the result set addressed with **INDEX**. The existing row in the result set is completely overwritten.

SQL UPDATE takes into account all columns entered in the "Select" command.

SQL
UPDATE

- ▶ **Parameter no. for result:** Q parameter in which the SQL server reports the result:
0: No error
1: Error occurred (incorrect handle, index too large, value range exceeded/fallen below or wrong data format)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set
Further Information: "SQL SELECT", page 270
- ▶ **Database: Index for SQL result:** Row number within the result set. The table entries prepared in the Q parameters are written to this row. If you do not specify the index, the first row (n=0) is written to.
Either enter the row number directly or program the Q parameter containing the index

Row number is transferred to a 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
```

Row number is programmed directly

```
...

40 SQL UPDATEQ1 HANDLE Q5 INDEX5
```

SQL INSERT

SQL INSERT generates a new row in the result set and transfers the data prepared in the Q parameters into the new row.

SQL INSERT takes into account all columns entered in the "Select" command. Table columns not entered in the "Select" command are filled with default values.

SQL
INSERT

- ▶ **Parameter no. for result:** Q parameter in which the SQL server reports the result:
0: No error
1: Error occurred (incorrect handle, value range exceeded/fallen below or wrong data format)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set
Further Information: "SQL SELECT", page 270

Row number is transferred to a Q parameter

```
11SQL BIND Q881
   "TAB_EXAMPLE.MEAS_NO"

12SQL BIND Q882
   "TAB_EXAMPLE.MEAS_X"

13SQL BIND Q883
   "TAB_EXAMPLE.MEAS_Y"

14SQL BIND Q884
   "TAB_EXAMPLE.MEAS_Z"

...

20SQL Q5 "SELECT
   MEAS_NO,MEAS_X,MEAS_Y,
   MEAS_Z FROM TAB_EXAMPLE"

...

40 SQL INSERTQ1 HANDLE Q5
```

Programming Q parameters

9.9 Accessing tables with SQL commands

SQL COMMIT

SQL COMMIT transfers all rows in the result set back to the table. A lock set with **SELECT...FOR UPDATE** is canceled.

The handle given in the **SQL SELECT** command loses its validity.



- ▶ **Parameter no. for result:** Q parameter in which the SQL server reports the result:
0: No error
1: Error occurred (incorrect handle or same entry in columns, in which unambiguous entries are required)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set
Further Information: "SQL SELECT", page 270

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 Q1HANDLE Q5 INDEX
	+Q2
...	
40	SQL UPDATEQ1 HANDLE Q5 INDEX
	+Q2
...	
50	SQL COMMITQ1 HANDLE Q5

SQL ROLLBACK

How **SQL ROLLBACK** is executed depends on whether **INDEX** is programmed:

- If **INDEX** is not programmed: The result set is **not** written back to the table (any changes/insertions are discarded). The transaction is closed and the handle given in the **SQL SELECT** command loses its validity. Typical application: Ending a transaction with read-access only
- If **INDEX** is programmed: The indexed row remains. All other rows are deleted from the result set. The transaction is **not** concluded. A lock set with **SELECT...FOR UPDATE** remains for the indexed row; for all other rows it is reset



- ▶ **Parameter no. for result:** Q parameter in which the SQL server reports the result:
0: No error
1: Error occurred (incorrect handle)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set
Further Information: "SQL SELECT", page 270
- ▶ **Database: Index for SQL result:** Row that is to remain in the result set. Either enter the row number directly or program the Q parameter containing the index














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 Q1HANDLE Q5 INDEX
	+Q2
...	
50	SQL ROLLBACKQ1 HANDLE Q5

9.10 Entering formulas directly

Entering formulas

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Press the **FORMULA** soft key to call the mathematical functions. The TNC 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. $Q12 = Q1 * (Q2 + Q3)$
	Square the value e.g. $Q15 = SQ 5$
	Square root e.g. $Q22 = SQRT 25$
	Sine of an angle e. g. $Q44 = SIN 45$
	Cosine of an angle e. g. $Q45 = COS 45$
	Tangent of an angle e. g. $Q46 = TAN 45$
	Arc sine Inverse function of the sine; determine the angle from the ratio of the opposite side to the hypotenuse e.g. $Q10 = ASIN 0.75$
	Arc cosine Inverse function of the cosine; determine the angle from the ratio of the adjacent side to the hypotenuse e. g. $Q11 = ACOS Q40$

Programming Q parameters

9.10 Entering formulas directly

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	Logarithmus Naturalis (LN) of a row 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	Truncate digits after the decimal point Form an integer e.g. Q3 = INT Q42
ABS	Absolute value of a number e. g. Q4 = ABS Q22
FRAC	Truncate digits before the decimal point Form a fraction e.g. Q5 = FRAC Q23
SGN	Check algebraic sign of a number e.g. Q12 = SGN Q50 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

12 Q1 = 5 * 3 + 2 * 10 = 35

- 1 Calculation $5 * 3 = 15$
- 2 Calculation $2 * 10 = 20$
- 3 Calculation $15 + 20 = 35$

or

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

$$a * (b + c) = a * b + a * c$$

Programming Q parameters

9.10 Entering formulas directly

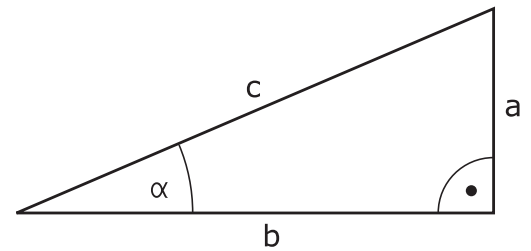
Example of entry

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.

Q ▶ Select the formula entry function: Press the **Q** key and the **FORMULA** soft key

FORMULA

Q ▶ Press the **Q** key on the external ASCII keyboard



PARAMETER NUMBER FOR RESULT?

ENT ▶ Enter **25** (parameter number) and press the **ENT** key

▶

▶ Shift the soft-key row and select the arc tangent function

ATAN

◁

▶ Shift the soft-key row and open the parentheses

(

Q

▶ Enter **12** (Q parameter number)

/

▶ Select division

Q

▶ Enter **13** (Q parameter number)

,

▶ Close parentheses and conclude formula entry

END
□

Example NC block

```
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 230

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	280
CFGREAD	Read out machine parameter	288
	Chain-linking string parameters	280
TOCHAR	Converting a numerical value to a string parameter	281
SUBSTR	Copy a substring from a string parameter	282
SYSSTR	Read out system parameter	283
Soft key	FORMULA string functions	Page
TONUMB	Converting a string parameter to a numerical value	284
INSTR	Checking a string parameter	285
STRLEN	Finding the length of a string parameter	286
STRCOMP	Compare alphabetic priority	287



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.

Programming Q parameters

9.11 String parameters

Assign string parameters

You have to assign a string variable before you use it. Use the **DECLARE STRING** command to do so.

SPEC
FCT

- Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- Open the function menu

STRING
FUNCTIONS

- Press the String functions soft key

DECLARE
STRING

- Press the **DECLARE STRING** soft key

Example NC block

```
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

- 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 TNC is to save the concatenated string. Confirm with the **ENT** key
- Enter the number of the string parameter in which the **first** substring is saved. Confirm with the **ENT** key: The TNC displays the concatenation symbol ||
- 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 TNC converts a numerical value to 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 | <ul style="list-style-type: none"> ► Select the function for converting a numerical value to a string parameter ► Enter the number or the desired Q parameter to be converted, and confirm with the ENT key ► If desired, enter the number of decimal places that the TNC should convert, and confirm with the ENT key ► Close the parenthetical expression with the ENT key and confirm your entry with the END key |

Example: Convert parameter Q50 to string parameter QS11, use 3 decimal places

```
37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```

Programming Q parameters

9.11 String parameters

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 TNC is to save the copied string. Confirm with the **ENT** key

SUBSTR

- ▶ Select the function for cutting out a substring
- ▶ Enter the number of the QS parameter from which the substring is to be copied. Confirm with the **ENT** key
- ▶ Enter the number of the place starting from which to copy the substring, and confirm with the **ENT** key
- ▶ Enter the number of characters to be copied, and confirm with the **ENT** key
- ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key



Remember that the first character of a text sequence starts internally with the zeroth place.

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2)

```
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
	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
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
	3	AFC control setting
	4	Tool-carrier kinematics

Programming Q parameters

9.11 String parameters

Converting a string parameter to a numerical value

The **TONUMB** function converts a string parameter to a numerical value. The value to be converted should be only numerical.



The QS parameter must contain only one numerical value. Otherwise the TNC will output an error message.

Q

- ▶ Select Q-parameter functions

FORMULA

- ▶ Press the **FORMULA** soft key
- ▶ Enter the number of the parameter in which the TNC is to save the numerical value. Confirm with the **ENT** key



- ▶ Shift the soft-key row

TONUMB










- ▶ Select the function for converting a string parameter to a numerical value
- ▶ Enter the number of the QS parameter to be converted, and confirm with the **ENT** key
- ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Convert string parameter QS11 to a numerical parameter Q82

```
37 Q82 = TONUMB ( SRC_QS11 )
```


Testing a string parameter

The **INSTR** function checks whether a string parameter is contained in another string parameter.

-  ▶ Select Q-parameter functions
-  ▶ Press the **FORMULA** soft key
-  ▶ Enter the number of the Q parameter for the result and confirm with the **ENT** key. The TNC saves in the parameter the position at which the sought-after text begins.
-  ▶ Shift the soft-key row
-  ▶ 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, and confirm with the **ENT** key
-  ▶ Enter the number of the place starting from which the TNC is to search the substring, and confirm with the **ENT** key
-  ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key



Remember that the first character of a text sequence starts internally with the zeroth place.

If the TNC 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 is found in more than one place, the TNC returns the first place at which it finds the substring.

Example: Search through QS10 for the text saved in parameter QS13. Begin the search at the third place.






```
37 Q50 = INSTR ( SRC_QS10 SEA_QS13 BEG2 )
```

Programming Q parameters

9.11 String parameters

Finding the length of a string parameter

The **STRLEN** function returns the length of the text saved in a selectable string parameter.

-  ▶ Select Q parameter function
-  ▶ Press the **FORMULA** soft key
-  ▶ Enter the number of the Q parameter in which the TNC is to save the ascertained string length. Confirm with the **ENT** key
-  ▶ Shift the soft-key row
- 
 - ▶ Select the function for finding the text length of a string parameter
 - ▶ Enter the number of the QS parameter whose length the TNC is to ascertain, and confirm with the **ENT** key
 - ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Find the length of QS15


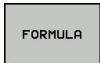
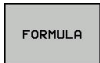





```
37 Q52 = STRLEN ( SRC_QS15 )
```



If the selected string parameter is not defined the control returns the result **-1**.

Compare alphabetic priority

The **STRCOMP** function compares string parameters for alphabetic priority.

-  ▶ Select Q parameter function
-  ▶ Press the **FORMULA** soft key
-  ▶ Enter the number of the Q parameter in which the TNC is to save the result of comparison. Confirm with the **ENT** key
-  ▶ Shift the soft-key row
-  ▶ Select the function for comparing string parameters
-  ▶ Enter the number of the first QS parameter to be compared, and confirm with the **ENT** key
-  ▶ Enter the number of the second QS parameter to be compared, and confirm with the **ENT** key
-  ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key



The TNC returns the following results:

- **0**: The compared QS parameters are identical
- **-1**: The first QS parameter **precedes** the second QS parameter alphabetically
- **+1**: The first QS parameter **follows** the second QS parameter alphabetically

Example: QS12 and QS14 are compared for alphabetic priority

```
37 Q52 = STRCOMP ( SRC_QS12 SEA_QS14 )
```





Programming Q parameters


9.11 String parameters

Reading out machine parameters

Use the **CFGREAD** function to read out TNC machine parameters as numerical values or as strings. The read values are always output in metric units.

In order to read out a machine parameter, you must use the TNC'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. To display the actual system names of the parameters, press the key for the screen layout key and then the **SHOW SYSTEM NAME** soft key. Follow the same procedure to return to the standard display.

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:

- ▶ Press the **Q** key.
- ▶ Press the **STRING FORMULA** soft key
- ▶ Enter the number of the string parameter in which the TNC is to save the machine parameter. Confirm with the **ENT** key
- ▶ Select the **CFGREAD** function
- ▶ Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the **ENT** key
- ▶ Enter the number for the index, or skip the dialog with **NNO ENT**, whichever applies
- ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Read as a string the axis designation of the fourth axis

Parameter settings in the configuration editor

```
DisplaySettings
CfgDisplayData
    axisDisplayOrder
        [0] to [3]
```

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

9.11 String parameters

Reading a numerical value of a machine parameter

Store the value of a machine parameter as a numerical value in a Q parameter:

- Q

FORMULA
- ▶ Select Q parameter function
 - ▶ Press the **FORMULA** soft key
 - ▶ Enter the number of the Q parameter in which the TNC is to save the machine parameter. Confirm with the **ENT** key
 - ▶ Select the **CFGREAD** function
 - ▶ Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the **ENT** key
 - ▶ Enter the number for the index, or skip the dialog with **NNO ENT**, whichever applies
 - ▶ Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Read overlap factor as Q parameter

Parameter settings in the configuration editor

ChannelSettings

CH_NC

CfgGeoCycle

pocketOverlap

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 TNC. The following types of information are assigned to Q parameters:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Results of measurements from touch probe cycles etc.

The TNC saves the values for the preassigned Q parameters Q108, Q114 and Q115 to Q117 in the unit of measure used by the active program.



Preassigned Q parameters (QS parameters) between **Q100** and **Q199** (**QS100** and **QS199**) must not be used in NC programs as calculation parameters. Otherwise you might receive undesired results.

Values from the PLC: Q100 to Q107

The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

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 TNC 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

Programming Q parameters

9.12 Preassigned Q parameters

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 overlap factor for pocket milling is assigned to Q112.

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 TNC 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 datum point 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 the TT 130

Deviation of actual from nominal value	Parameter value
Tool length	Q115
Tool radius	Q116

10

**Miscellaneous
functions**

Miscellaneous functions

10.1 Enter miscellaneous functions M

10.1 Enter miscellaneous functions M

Fundamentals

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



If several M functions are programmed in one NC block, then 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 take effect at the start or end of block, execution take place in the sequence 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 NC blocks

87 STOP M6

Miscellaneous functions

10.2 Miscellaneous functions for program run inspection, spindle and coolant

10.2 Miscellaneous functions for program run inspection, spindle and coolant

Overview



The machine tool builder can influence the behavior of the miscellaneous functions described below. Refer to your machine manual.

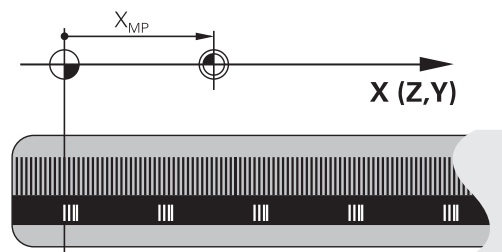
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope depends on machine parameter clearMode (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 datum

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 TNC references coordinates to the workpiece datum.

Further Information: "Datum setting without a 3-D touch probe", page 359

Behavior with M91 – Machine datum

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



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

The coordinate values on the TNC screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF.

Further Information: "Status displays", page 85

Miscellaneous functions

10.3 Miscellaneous functions for coordinate entries

Behavior with M92 – Additional machine datum



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

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

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



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

Effect

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

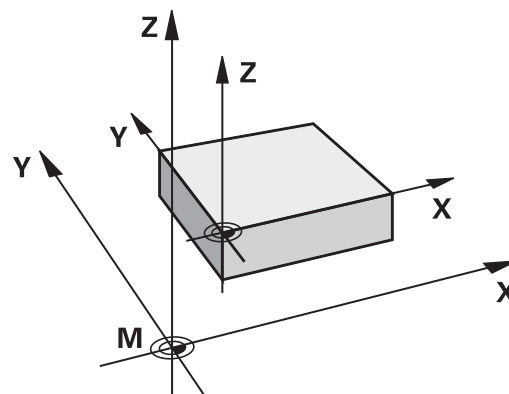
M91 and M92 take effect at the start of block.

Workpiece datum

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

If datum setting is inhibited for all axes, the TNC no longer displays the **DATUM SET** 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 set datum.

Further Information: "Showing the workpiece blank in the working space", page 399

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

Standard behavior

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

Example:

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

Behavior with M94

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

Example NC blocks

To reduce display of all active rotary axes:

```
M94
```

To reduce display of the C axis only:

```
M94 C
```

To reduce 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 in which it is programmed. M94 becomes effective at the start of block.

Miscellaneous functions

10.4 Miscellaneous functions for path behavior

10.4 Miscellaneous functions for path behavior

Feed rate factor for plunging movements: M103

Standard behavior

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

Behavior with M103

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

$$FZMAX = FPROG \times F\%$$

Programming M103

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

Effect

M103 becomes effective at the start of block.

To cancel M103, program M103 once again without a factor.

Feed rate in millimeters per spindle revolution: M136

Standard behavior

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

Behavior with M136



In inch-programs, M136 is not permitted in combination with the new alternate feed rate FU.
The spindle is not permitted to be controlled when M136 is active.

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

Effect

M136 becomes effective at the start of block.

You can cancel M136 by programming M137.

Miscellaneous functions

10.4 Miscellaneous functions for path behavior

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

Standard behavior

In the **Program run single block** and **Program run full sequence** modes, the TNC moves the tool as defined in the machining program.

Behavior with M140

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

Input

If you enter M140 in a positioning block, the TNC continues the dialog and asks for the desired path of tool departure from the contour. Enter the requested path that the tool should follow when departing the contour, or press the 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 TNC moves the tool along the entered path at rapid traverse.

Effect

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

M140 becomes effective at the start of the block.

Example NC blocks

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

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

```
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

Special functions

11.1 Overview of special functions

11.1 Overview of special functions

The TNC provides the following powerful special functions for a large number of applications:

Function	Description
Working with text files	page 323
Working with freely definable tables	page 309

Press the **SPEC FCT** and the corresponding soft keys to access further special functions of the TNC. The following tables give you an overview of which functions are available.

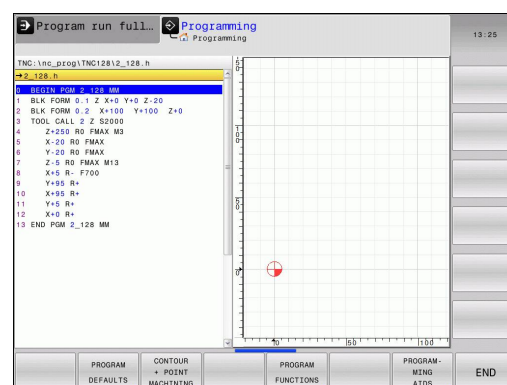
Main menu for SPEC FCT special functions

- SPEC FCT** ▶ Press the SPEC FCT key to select the special functions

Soft key	Function	Description
PROGRAM DEFAULTS	Define program defaults	page 307
CONTOUR + POINT MACHINING	Functions for contour and point machining	page 307
PROGRAM FUNCTIONS	Define different conversational functions	page 308
PROGRAM- MING AIDS	Programming aids	page 151



After pressing the **SPEC FCT** key, you can open the **smartSelect** selection window with the **GOTO** key. The TNC 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 TNC displays online help for the specific functions in the window on the right.

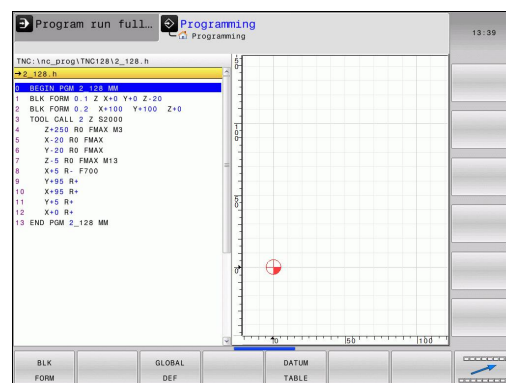


Program defaults menu

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 538
GLOBAL DEF	Define global cycle parameters	page 458

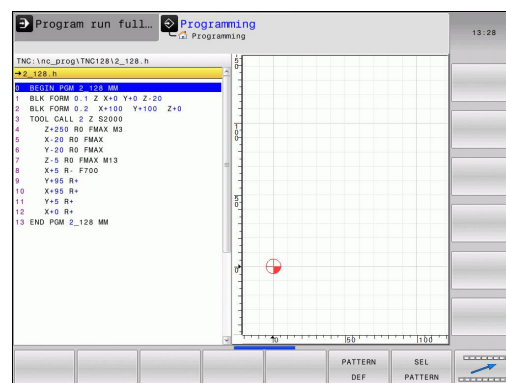


Functions for contour and point machining menu

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 462
SEL PATTERN	Select the point file with machining positions	page 473



Special functions

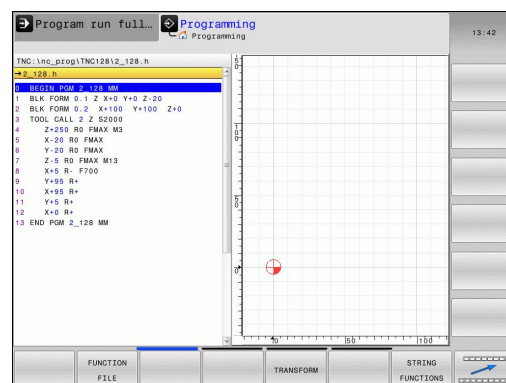
11.1 Overview of special functions

Menu of various conversational functions

PROGRAM
FUNCTIONS

- Press the soft key for defining various conversational functions

Soft key	Function	Description
FUNCTION FILE	Define file functions	page 319
TRANSFORM	Define coordinate transformations	page 320
STRING FUNCTIONS	Define string functions	page 279
FUNCTION SPINDLE	Define pulsing spindle speed	page 315
FUNCTION FEED	Define dwell time	page 317
INSERT COMMENT	Add comments	page 153



11.2 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.

NR	X	Y	Z	A	C	DOC
0	100.001	49.999	0			PAT 1
1	99.994	49.999	0			PAT 2
2	99.999	50.001	0			PAT 3
3	100.002	49.995	0			PAT 4
4	99.990	50.003				PAT 5
5						
6						
7						
8						
9						
10						

Creating a freely definable table

- ▶ To call the file manager, press the **PGM MGT** key
- ▶ Enter any file name with the .TAB extension and confirm with the **ENT** key. The TNC displays a pop-up window with permanently saved table formats
- ▶ Use the arrow key to select a table template e.g. **EXAMPLE.TAB** and confirm with the **ENT** key: The TNC 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 310



Machine tool builders may define their own table templates and save them in the TNC. When you create a new table, the TNC opens a pop-up window listing all available table templates.



You can also save your own table templates in the TNC. To do this, you 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.

Special functions

11.2 Freely definable tables

Editing the table format

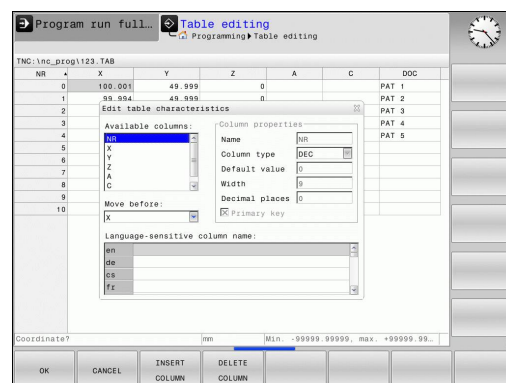
- Press the soft key **EDIT FORMAT** (switch the soft-key row):
The TNC opens the editor form showing the table structure.
The meanings of the structure commands (header entries) are shown in the following table.

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 UPTXT: 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

You can use a connected mouse or the TNC keyboard to navigate in the form. Navigation using the TNC 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.

In a field of the **TSTAMP** column type you can reset an invalid value if you press the **CE** key and then the **ENT** key.

Exiting the structure editor

- Press the **OK** soft key. The TNC 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.

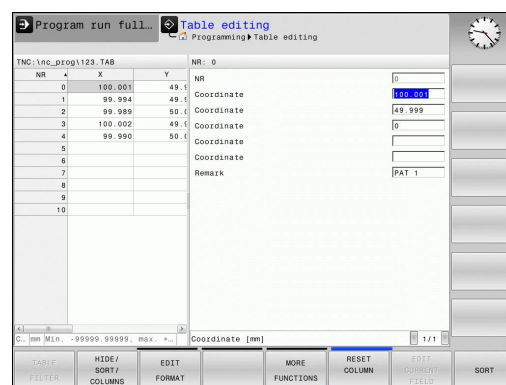


- Press the key for setting the screen layout. Select the respective soft key for list view or form view (form view: with or without dialog texts)

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

In the right half you can change the data.

- Press the **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.



Special functions

11.2 Freely definable tables

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 write several column names in a **TABWRITE** block. The column names must be written between quotation marks and separated by a comma. You define the values that the TNC is to write to the respective column with Q parameters.



Note that by default the **FN 27: TABWRITE** function also writes values to the currently open table in **Test run** mode. The **FN18 ID992 NR16** function enables you to query in which operating mode the program is to be run. 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 command.

Further Information: "If-then decisions with Q parameters", page 240

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
```

Special functions

11.2 Freely definable tables

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, several 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 TNC 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 TNC 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"
```

Customize table view



This function may only be used with the permission of your machine manufacturer.

Soft key

Function

ADAPT THE
TABLE
FORMAT

Adapt format of tables present after changing the control software version

11.3 Pulsing spindle speed FUNCTION S-PULSE

Program pulsing spindle speed

Application



The machine manual provides further information.

- Function is machine-dependent
- Possible hazards for the operator and the machine
- ▶ Follow the safety precautions and the functional description of the machine tool builder.

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 the SCALE input value. The spindle speed changes in a sinusoidal form around the target value.

Procedure

Proceed as follows for the definition:

- | | |
|----------------------|---|
| SPEC
FCT | ▶ Show the soft-key row with special functions |
| PROGRAM
FUNCTIONS | ▶ Select the menu for defining various conversational functions |
| 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.

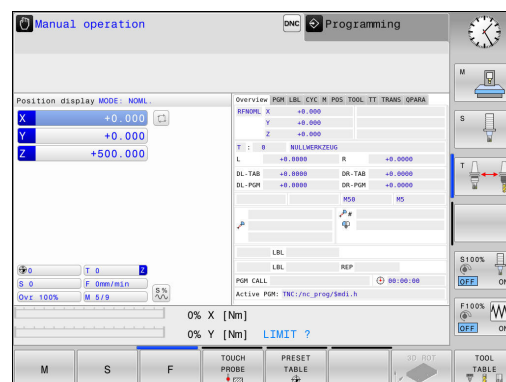
NC block

**13 FUNCTION S-PULSE P-TIME10
SCALE5**

Symbols

In the status bar the symbol indicates the condition of the pulsing shaft speed:

Icon	Function
	Pulsing spindle speed active



Special functions

11.3 Pulsing spindle speed FUNCTION S-PULSE

Reset pulsing spindle speed

Use the **S-PULSE RESET** function to reset the pulsing spindle speed.

Proceed as follows for the definition:

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Select the menu for defining various plain-language functions

FUNCTION
SPINDLE

- ▶ Press the **FUNCTION SPINDLE** soft key

RESET
SPINDLE-
PULSE

- ▶ Press the **RESET SPINDLE-PULSE** soft key.

NC block

18 FUNCTION S-PULSE RESET

11.4 Dwell time FUNCTION FEED

Programming dwell time

Application



The machine manual provides further information.

- Function is machine-dependent
- Possible hazards for the operator and the machine
- Follow the safety precautions and the functional description of the machine tool builder.

The **FUNCTION FEED DWELL** function is 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.

Damage to the workplace!

- Do not use **FUNCTION FEED DWELL** to manufacture threads

Procedure

Proceed as follows for the definition:

SPEC
FCT

- Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- Select the menu for defining various plain-language functions

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

NC block

13 FUNCTION FEED DWELL D-TIME0.5
F-TIME5

Special functions

11.4 Dwell time FUNCTION FEED

Resetting dwell time



Reset to the dwell time immediately following the machining with chip breaking.

NC block

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

- Select the menu for defining various plain-language functions

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 TNC automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

11.5 File functions

Application

The **FILE FUNCTION** features are used to copy, move and delete files from within the part 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 the file functions: The TNC 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

Special functions

11.6 Definition of a datum shift






11.6 Definition of a datum shift

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, which you can use to easily reset a datum shift.

TRANS DATUM AXIS

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:

- 
 - ▶ Show the soft-key row with special functions
- 
 - ▶ Select the menu for defining various conversational functions
- 
 - ▶ Select transformations
- 
 - ▶ Select datum shifting with **TRANS DATUM**
- 
 - ▶ 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 values refer to the workpiece datum, which is specified either by datum setting or with 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).

NC block

13 TRANS DATUMAXIS X+10 Y+25 Z+42

TRANS DATUM TABLE

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:

- 
 - ▶ Show the soft-key row with special functions
- 
 - ▶ Select the menu for defining various plain-language functions
- 
 - ▶ Select transformations
- 
 - ▶ Select datum shifting with **TRANS DATUM**
- 
 - ▶ Select datum shifting with **TRANS DATUM TABLE**
 - ▶ Enter the line number to be activated by the TNC, and 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 did not define a datum table in the **TRANS DATUM TABLE** block, then the TNC uses the datum table already selected in the NC program with **SEL TABLE**, or the datum table with status M selected in the **Program run, single block** or **Program run, full sequence** operating mode.

NC block

13 TRANS DATUMTABLE TABLINE25

Special functions

11.6 Definition of a datum shift

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:

NC block

13 TRANS DATUM RESET

SPEC
FCT

- ▶ Show the soft-key row with special functions

PROGRAM
FUNCTIONS

- ▶ Select the menu for defining various conversational functions

TRANSFORM

- ▶ Select transformations

TRANS
DATUM

- ▶ Select datum shift **TRANS DATUM**

RESET
DATUM
SHIFT

- ▶ Press the **RESET DATUM SHIFT** soft key

11.7 Creating text files

Application

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




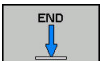
- Recording test results
- Documenting working procedures
- Creating formula collections

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

Special functions

11.7 Creating text files

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.

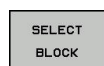
- ▶ Move the cursor to the word or line that you wish to erase and insert at a different place in the text
- ▶ Press the **DELETE WORD** or **DELETE LINE** soft key: The text is deleted and stored temporarily.
- ▶ Move the cursor to the location where you wish insert the text, and press the **INSERT LINE / WORD** soft key.

Soft key	Function
	Delete and temporarily store a line
	Delete and temporarily store a word
	Delete and temporarily store a character
	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.



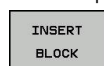
- ▶ Press the **SELECT BLOCK** soft key.
- ▶ Move the cursor to the last character of the text you wish to select. You can select whole lines by moving the cursor up or down directly with the arrow keys—the selected text is shown in a different color.

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

Soft key	Function
	Delete the selected block and store temporarily
	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

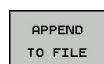


- ▶ Press the **INSERT BLOCK** soft key the text block is inserted.

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

Transferring the selected block to a different file

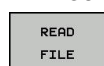
- ▶ Select the text block as described previously



- ▶ Press the **APPEND TO FILE** soft key. The TNC displays the dialog prompt **file name**
- ▶ Enter the path and name of the destination file. The TNC appends the selected text 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



- ▶ Press the **READ FILE** soft key. The TNC displays the dialog prompt **File name =**
- ▶ Enter the path and name of the file you want to insert

Special functions

11.7 Creating text files

Finding text sections

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

Finding the current text

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

- ▶ Move the cursor to the desired word.
- ▶ To select the search function, press the **FIND** soft key.
- ▶ Press the **FIND CURRENT WORD** soft key.
- ▶ To 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 TNC displays the dialog prompt **Find text :**
- ▶ 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

11.8 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 factor 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!

Special functions

11.8 Tool carrier management










Assign 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**.

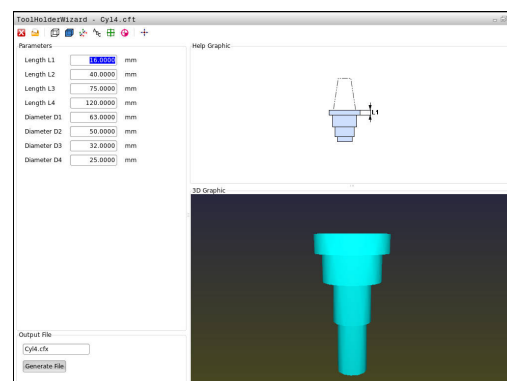
The additional **ToolHolderWizard** tool is mainly operated with a mouse. Using the mouse, you can also set the desired screen layout by drawing a line between the areas **Parameter**, **Help graphics** and **3-D graphics** by holding down the left mouse button.

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



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.



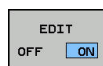
Proceed as follows to parameterize tool carrier templates and save these parameters:



- ▶ Operating mode: Press the **MANUAL OPERATION** key



- ▶ Press the **TOOL TABLE** soft key



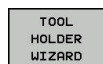
- ▶ Press the **EDIT** soft key



- ▶ Move the cursor to the **KINEMATIC** column



- ▶ 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



11.8 Tool carrier management

Allocate 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**.



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!

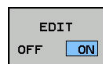
Proceed as follows to allocate a parameterized tool carrier to a tool:



- ▶ Operating mode: Press the **MANUAL OPERATION** key



- ▶ Press the **TOOL TABLE** soft key



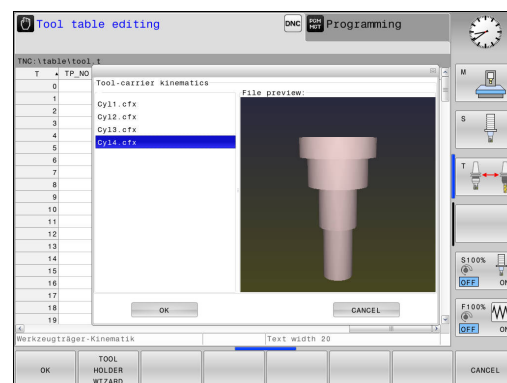
- ▶ Press the **EDIT** soft key



- ▶ 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



11.9 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

Proceed as follows for the definition:

- ▶ Show the soft-key row with special functions
- ▶ Select the menu for defining various plain-language functions
- ▶ **FUNCTION DWELL** soft key
- ▶ Press the **DWELL TIME** soft key
- ▶ Define the duration in seconds
- ▶ Alternatively, press the **DWELL REVOLUTIONS** soft key
- ▶ Define the number of spindle revolutions

NC block

13 FUNCTION DWELL TIME10

NC block

23 FUNCTION DWELL REV5.8

12

**Manual Operation
and Setup**

Manual Operation and Setup

12.1 Switch-on, switch-off

12.1 Switch-on, switch-off

Switch-on



Refer to your machine manual.

Danger exists for the operator when the machine is started up. Read the safety information before switching on the machine.



Switch-on and crossing over the reference points can vary depending on the machine tool.

Refer to your machine manual.

Switch on the power supply for TNC and machine. The TNC then displays the following dialog:

SYSTEM STARTUP

- ▶ TNC is started

POWER INTERRUPTED



- ▶ TNC message that the power was interrupted—clear the message

COMPILE A PLC PROGRAM

- ▶ The PLC program of the TNC is automatically compiled

RELAY EXT. DC VOLTAGE MISSING



- ▶ Switch on external dc voltage. The TNC checks the functioning of the EMERGENCY STOP circuit

MANUAL OPERATION

TRAVERSE REFERENCE POINTS



- ▶ Cross the datums manually in the prescribed sequence: For each axis press the **NC START** key; or



- ▶ Cross the datums in any sequence: Press and hold the machine axis direction button for each axis until the datum has been traversed



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

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



The reference points need only be crossed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the **Programming** or **Test run** mode of operation immediately after switching on the control voltage. You can cross the reference points later. For this purpose, in **Manual operation** mode press the **PASS OVER REFERENCE** soft key.

Switch-off

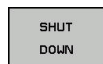


Deactivation is a machine-dependent function. Refer to your machine manual.

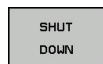
To prevent data from being lost on switch-off, you need to shut down the operating system of the TNC as follows:



- ▶ Operating mode: Press the **Manual operation** key



- ▶ Select the function for shutting down



- ▶ Confirm with the **SHUT DOWN** soft key
- ▶ When the TNC displays the message **Now you can switch off the TNC** in a pop-up window, you may switch off the power supply to the TNC



Caution: Data may be lost!

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

The control restarts after pressing the **RESTART** soft key. Switch-off during a restart can also result in data loss!

12.2 Moving the machine axes

Note



Refer to your machine manual.

Moving with 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 349

If a moving task is active on the machine, the control displays the **control in operation** symbol.

Incremental jog positioning

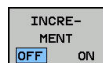
With incremental jog positioning you can move a machine axis by a preset distance.



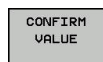
- ▶ Operating mode: Press the **MANUAL OPERATION** or **ELECTRONIC HANDWHEEL** key



- ▶ Shift the soft-key row



- ▶ Select incremental jog positioning: Switch the **INCREMENT** soft key to **ON**



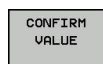
- ▶ Enter the infeed of the **linear axes** and confirm with the **CONFIRM VALUE** soft key



- ▶ Alternatively, confirm with the **ENT** key



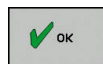
- ▶ Use the arrow keys to position the cursor on the **rotary axis**



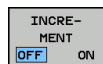
- ▶ Enter the infeed of the **rotary axes** and confirm with the **CONFIRM VALUE** soft key



- ▶ Alternatively, confirm with the **ENT** key



- ▶ Confirm with the **OK** soft key
- ▶ The increment is active.

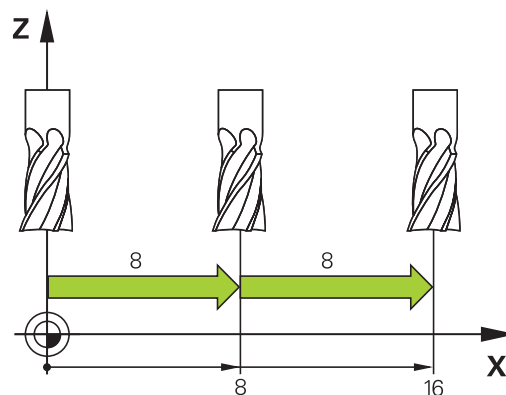


- ▶ Deactivate incremental jog positioning: Switch the **INCREMENT** soft key to **OFF**



If you are in the increment menu, you can switch off incremental jog positioning with the **SWITCH OFF** soft key.

The maximum permissible value for infeed is 10 mm.



12

Manual Operation and Setup

12.2 Moving the machine axes

Traverse with the HR 410 electronic handwheel

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

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







The HR 410 handwheel features the following operating elements:

- 1 EMERGENCY STOP 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 TNC moves the selected axis
- 7 Machine functions (set by the machine manufacturer)



Traversing axes

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

-  ▶ Select the **Electronic handwheel** operating mode
-  ▶ Press and hold a permissive button
-  ▶ Select the axis
-  ▶ Select the feed rate
-  ▶ Move the active axis in the positive direction, or
-  ▶ Move the active axis in the negative direction

Moving with the electronic display handwheels

The TNC supports traversing with the following new electronic handwheels:

- HR 520: Handwheel with display, data transfer per cable
- HR 550FS: Handwheel with display, data transfer via radio



Caution: Danger to the operator and handwheel!

All of the handwheel connectors may only be removed by authorized service personnel, even if it is possible without any tools!

Ensure that the handwheel is plugged in before you switch on the machine!

If you wish to operate your machine without the handwheel, disconnect the cable from the machine and secure the open socket with a cap!



Refer to your machine manual. Your machine tool builder can make additional functions of the HR 5xx available.

The portable HR 5xx handwheels feature a display on which the TNC shows information. In addition, you can use the handwheel soft keys for important setup functions, e.g. datum setting or entering and running M functions.

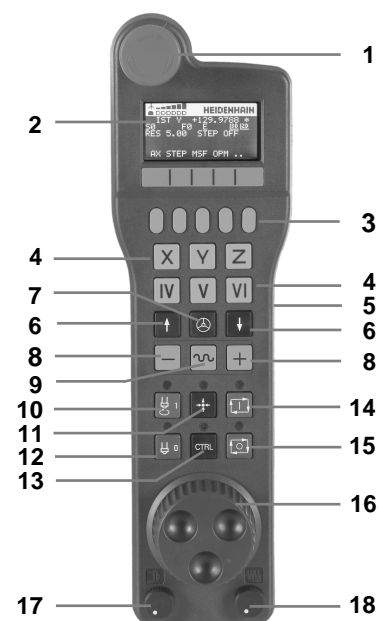
As soon as you have activated the handwheel with the handwheel activation key, the operating panel is locked. This is indicated by a pop-up window on the TNC screen.



Manual Operation and Setup

12.2 Moving the machine axes

- 1 EMERGENCY STOP** key
- 2** Handwheel display for status and for selecting functions
- 3** Soft Keys
- 4** Axis selection 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 TNC 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
- 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 TNC additionally displays the current traversing step
- 12 Soft-key row:** Selection of various functions, described in the following sections



12.2 Moving the machine axes

Special features of the wireless handwheel HR 550FS



Due to various potential sources of interference, a wireless connection is not as reliable as a cable connection. Before you use the wireless handwheel it must therefore be checked whether there are any other radio users in the surroundings of the machine. This inspection for presence of radio frequencies or channels is recommended 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.

**Caution: Danger to the operator and machine!**

Due to safety reasons you must switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the TNC can run a functional test when it is restarted!

If you use several machines with wireless handwheels in your workshop you have to mark the handwheels and holders that belong together so that their respective associations are clearly identifiable (e.g. by color stickers or numbers). The markings on the wireless handwheel and the handwheel holder must be clearly visible to the user!

Before every use, make sure that the correct handwheel for your machine is active.



Moving the machine axes 12.2

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.

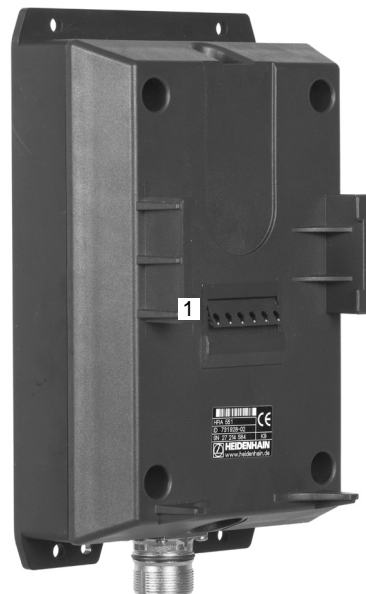
Clean the contacts **1** in the handwheel holder and of the handwheel regularly to ensure their proper functioning.

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.



Caution: Danger to the workpiece and tool!

If interruption-free operation is no longer possible within the transmission range the TNC automatically triggers an emergency stop. This can also happen during machining. Keep the distance to the handwheel holder to a minimum. If you do not use the handwheel, put it in the handwheel holder.



Manual Operation and Setup

12.2 Moving the machine axes

If the TNC has triggered an emergency stop you must reactivate the handwheel. Proceed as follows:

- ▶ Select the **Programming** operating mode
- ▶ Press the **MOD** key to select the MOD function
- ▶ Scroll through the soft-key row
 - ▶ 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 the **END** button

The **MOD** operating mode includes a function for commissioning and configuring the handwheel.

Further Information: "Configuring the HR 550FS wireless handwheel", page 446

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 address keys. Your machine manufacturer 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 selection keys, proceed as follows:

- ▶ Press the handwheel soft key **F1 (AX)**: The TNC displays 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



- ▶ Activate the handwheel: Press the handwheel key on the HR 5xx: Now you can only operate the TNC via the HR 5xx; the TNC shows a pop-up window containing information on the TNC 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



- ▶ Deactivate the handwheel: Press the handwheel key on the HR 5xx: Now you can operate the TNC again via the operating panel

Potentiometer settings

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

- ▶ Press the **CTRL** and handwheel keys on the HR 5xx; the TNC shows the soft-key menu for selecting the potentiometers on the handwheel display
- ▶ Press the **HW** soft key to activate the handwheel potentiometers

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

- ▶ Press the **CTRL** and handwheel keys on the HR 5xx; the TNC shows the soft-key menu for selecting the potentiometers on the handwheel display
- ▶ Press the **KBD** soft key to activate the potentiometers of the machine operating panel

12.2 Moving the machine axes

Incremental jog positioning

With incremental jog positioning the TNC 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. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. By also pressing the **CTRL** key, you can increase the counting increment to 1. 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

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. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. By also pressing the **CTRL** key, you can increase the counting increment to 1000
- ▶ Activate the new speed S with the **NC START** key

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. If you press and hold the respective key, each time it reaches a decimal value 0 the TNC increases the counting increment by a factor of 10. By also pressing the **CTRL** key, you can increase the counting increment to 1000
- ▶ Load the new feed rate F with the **F3 (OK)** handwheel soft key

Setting a datum

- ▶ Press handwheel soft key **F3 (MSF)**
- ▶ Press the handwheel soft key **F4 (PRS)**
- ▶ If required, select the axis in which the datum is to be set
- ▶ 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 with manl.data input**
 - SGL: **Program run, single block**
 - RUN: **Program run, full sequence**

12.2 Moving the machine axes

Generating a complete traversing block



Your machine tool builder can assign any function to the "Generate NC block" handwheel key. Refer to your machine manual.

- ▶ Switch to the **Positioning with manl.data input** mode of operation
- ▶ If required, use the arrow keys on the TNC keyboard to select the NC block after which the new traversing block is to be inserted.
- ▶ Activate the handwheel
- ▶ Press the "Generate NC block" handwheel key: The TNC 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 416
- 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 296



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

Entering values

Spindle speed S, miscellaneous function M



- Select input for spindle speed: press the **S** soft key

SPINDLE SPEED S=



- Enter **1000** (spindle speed) and load 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.

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 lowest feed rate from the machine parameter **manualFeed** (no. 400304) takes effect
- If the feed rate entered exceeds the value defined in the machine parameter **maxFeed** (no. 400302) then the parameter value in the machine parameter takes effect
- F is not lost during a power interruption
- The control displays the feed rate.

Manual Operation and Setup

12.3 Spindle speed S, feed rate F and miscellaneous function M

Adjusting spindle speed and feed rate

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

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



The override knob 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 with manl.data input

Procedure

To activate the feed rate limit F MAX, proceed as follows:



- ▶ Operating mode: Press the **POSITIONING WITH MANL.DATA INPUT** key



- ▶ Press the **F MAX** soft key



- ▶ Enter the desired maximum feed rate
- ▶ Press the OK soft key

Datum management with the preset table 12.4

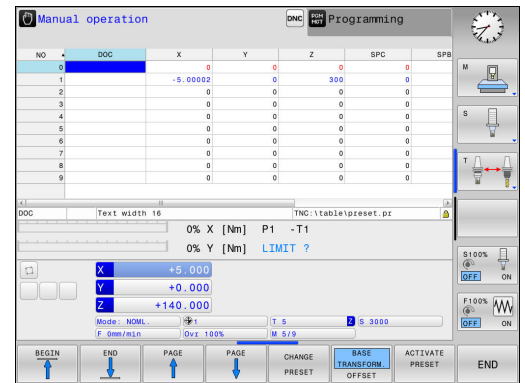
12.4 Datum management with the preset table

Note



You should definitely use the preset table if:

- Up to now you have been working with older TNC controls with REF-based datum tables



The preset table can contain any number of lines (datums). To optimize the file size and the processing speed, only use as many rows as you need to manage your datums.

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

12.4 Datum management with the preset table

Saving the datums in the preset table

The preset table has the name **PRESET.PR**, and is saved in the directory **TNC:\table**. **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.

It is permitted to copy the preset table into another directory (for data backup). 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 directory **TNC:\table**.

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


- Manual input
- Using the probing cycles in the **MANUAL OPERATION** and **ELECTRONIC HANDWHEEL** modes











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

Manually saving the datums in the preset table

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



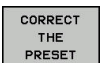


- 
 - ▶ Operating mode: Press the **Manual operation** key

- 
 - ▶ Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly
- 
- 


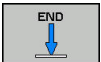



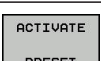

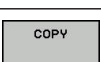

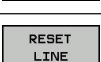
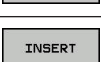
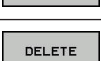
- 
 - ▶ Press the **PRESET TABLE** soft key
 - ▶ The TNC opens the preset table and sets the cursor to the row of the active datum.
- 
 - ▶ Select the functions for preset entry
 - ▶ The TNC displays all available input options in the soft-key row.
- 
 - ▶ Select the row in the preset table that you want to change (the row number is the preset number)
- 
 - ▶ If needed, select the column (axis) in the preset table that you want to change
- 
 - ▶ Use the soft keys to select one of the available entry possibilities

12.4 Datum management with the preset table

Input options

Soft key	Function
	Directly transfer the actual position of the tool (the measuring dial) as the new datum: This function only saves the preset in the axis in which the cursor is currently hovering.
	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
	Incrementally shift a datum 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 TNC will internally convert the entered values to mm
	Directly enter the new datum without calculation of the kinematics (axis-specific). Only use this function if your machine has a rotary table, and you want to set the datum to the center of the rotary table by entering 0. This function only saves the preset 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 TNC will internally convert the entered values to mm
	Write the currently active datum to a selectable line in the table: This function saves the datum in all axes, and then activates the appropriate row in the table automatically. If inch display is active: Enter the value in inches, and the TNC will internally convert the entered values to mm

Editing the preset table

Soft key	Editing function in table mode
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Select the functions for entering presets
	Activate the datum of the selected line of the preset table
	Add the entered number of lines to the end of the table (2nd soft-key row)
	Copy the highlighted field (2nd soft-key row)
	Insert the copied field (2nd soft-key row)
	Reset the selected line: The TNC enters - in all columns (2nd soft-key row)
	Insert a single line at the end of the table (2nd soft-key row)
	Delete a single line at the end of the table (2nd soft-key row)

12.4 Datum management with the preset table

Overwrite protection for datum

Row 0 in the preset table is write-protected. The TNC saves the last manually set datum in row 0.

You can protect further 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).

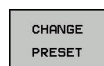
**Caution: Data may be lost!**

If you forget the password, then you can no longer reset the write protection in a protected row.

If you protect a row with a password, please make a note of this password.

Ideally, use simple protection with the **LOCK / UNLOCK** soft key.

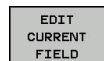
Proceed as follows to protect a datum from overwriting:



- ▶ Press the **CHANGE PRESET** soft key

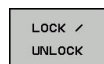


- ▶ Select the **LOCKED** column



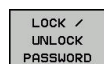
- ▶ Press the **EDIT CURRENT FIELD** soft key

Protection for datum without using password:

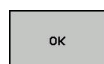


- ▶ Press the **LOCK / UNLOCK** soft key
- > The TNC writes an **L** in the **LOCKED** column.

Protect a datum with a password:



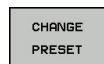
- ▶ Press the **LOCK / UNLOCK PASSWORD** soft key



- ▶ Enter the password into the pop-up window
- ▶ Confirm with the **OK** soft key or with the **ENT** key:
- > The TNC writes **###** to the **LOCKED** column.

Rescind write-protection

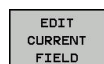
To edit a line you have previously write-protected, proceed as follows:



- ▶ Press the **CHANGE PRESET** soft key

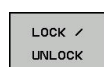


- ▶ Select the **LOCKED** column



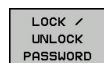
- ▶ Press the **EDIT CURRENT FIELD** soft key

Datum protected without password:

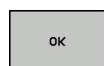


- ▶ Press the **LOCK / UNLOCK** soft key
- > The TNC rescinds the write protection.

Datum protected with a password:



- ▶ Press the **LOCK / UNLOCK PASSWORD** soft key



- ▶ Enter the password into the pop-up window
- ▶ Confirm with the **OK** soft key or with the **ENT** key
- > The TNC rescinds the write protection.

12.4 Datum management with the preset table

Activating the datum

Activating a datum from the preset table in the Manual operation mode



When activating a datum from the preset table, the TNC resets the active datum shift, mirroring and scaling factor.



- ▶ Operating mode: Press the **Manual operation** key



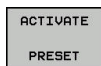
- ▶ Display the preset table: Press the **PRESET TABLE** soft key



- ▶ Select the datum number that you want to activate; or



- ▶ With the **GOTO** key, select the datum number that you want to activate and confirm with the **ENT** key



- ▶ Activate the datum: Press the **ACTIVATE PRESET** soft key



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



- ▶ Exit the preset table

Activating a datum from the preset table in an NC program

Use Cycle 247 in order to activate datums from the preset table during a program run. In Cycle 247 you simply define the number of the datum to be activated.

Further Information: "DATUM SETTING (Cycle 247)", page 543

12.5 Datum setting without a 3-D touch probe

Note

When you set a datum, you set the TNC display to the coordinates of a known workpiece position.



All manual probe functions are available with a 3-D touch probe.

Further Information: "Datum setting with a 3-D touch probe (option number 17)", page 375

Preparation

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

Datum setting with an end mill



Protective measure

If the workpiece surface must not be scratched, you can lay a metal shim of known thickness d on it. You then enter a value that is greater than the desired preset by the value d .



- ▶ Operating mode: Press the **Manual operation** key



- ▶ Move the tool slowly until it touches (scratches) the workpiece surface



- ▶ Select the axis

DATUM SETTING Z=



- ▶ 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

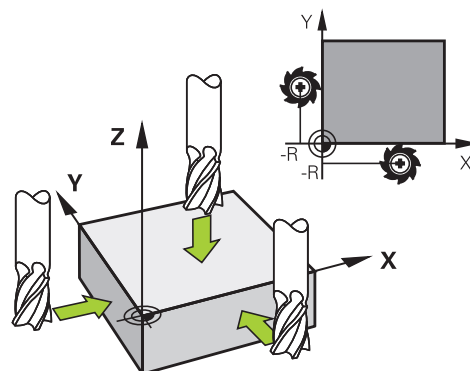


Repeat the process for the remaining axes.

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 $Z=L+d$.



The TNC automatically saves the datum set with the axis keys in line 0 of the preset table.



12.5 Datum setting without a 3-D touch probe

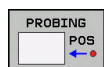
Using touch probe functions with mechanical probes or measuring dials

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 361

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 TNC
- ▶ Confirm the position: Press the **ACTUAL-POSITION CAPTURE** soft key for the TNC to save the current position
- ▶ Move the mechanical probe to the next position to be captured by the TNC
- ▶ Confirm the position: Press the **ACTUAL-POSITION CAPTURE** soft key for the TNC to save the current position
- ▶ If required, move to additional positions and capture as described previously
- ▶ **Datum:** In the menu window, enter the coordinates of the new datum, confirm with the **SET DATUM** soft key, or write the values to a table
- ▶ **Further Information:** "Writing measured values from the touch probe cycles to a datum table", page 367
- ▶ **Further Information:** "Writing measured values from the touch probe cycles to the preset table", page 368
- ▶ Terminate the probing function: Press the **END** key

12.6 Using a 3-D touch probe (option 17)

Overview

The following touch probe cycles are available in the **Manual operation** mode:








HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Make sure during probing that the axis angles match the set tilt angles. The control checks this automatically if the **chkTiltingAxes** machine parameter (no. 204601) is activated.



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

The touch probe cycles are available only with option number 17. If you are using a HEIDENHAIN touch probe, this option is available automatically.

Soft key	Function	Page
	Calibrating the 3-D Touch Probe	369
	Setting the datum on any axis	375
	Set a circle center as preset	376
	Setting the centerline as datum	379
	Touch probe system data management	562

12.6 Using a 3-D touch probe (option 17)

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:

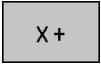
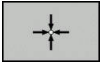
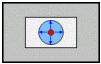
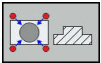

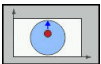
- ▶ Start the manual touch probe cycle
- ▶ Position the touch probe at a position near the first touch point
- ▶ Probe the first touch point
- ▶ Activate the handwheel on the handwheel
- > The control shows the pop-up window **Handwheel active**.
- ▶ Position the touch probe at a position near the second touch point
- ▶ Deactivate the handwheel on the handwheel
- > The control closes the pop-up window.
- ▶ Probe the second touch point
- ▶ If necessary, set the datum
- ▶ End the probing function



If the handwheel is active you cannot start the probing cycles.

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
	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

Manual Operation and Setup

12.6 Using a 3-D touch probe (option 17)

Automatic probing routine for holes, studs and model circles



If you use a function for probing a circle automatically, the TNC automatically positions the touch probe to the respective touch points. Ensure that the positions can be approached without collision.

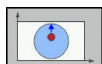
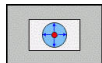
If you use a probing routine for automatically probing a hole or a stud, or a model circle, the control opens a form with the required entry fields.

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 TNC carries out all pre-positioning and probing processes automatically

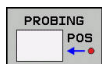


The TNC approaches the position at the feed rate **FMAX** defined in the touch probe table. The defined probing feed rate **F** is used for the actual probing operation.



Before starting the automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance (value from the touch probe table + value from the input form) opposite to the probing direction.

Selecting the probing cycle



- ▶ Operating mode: Select **Manual operation** or **Electronic handwheel**
- ▶ 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**, for the TNC to display the associated menu



When you select a manual probing function, the TNC 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 move to the desired input field. You can position the cursor only in fields that can be edited. Fields that cannot be edited are shown gray.

12.6 Using a 3-D touch probe (option 17)

Recording measured values from the touch probe cycles



The TNC must be specially prepared by the machine tool builder for use of this function. Refer to your machine manual.

After executing the respective selected touch probe cycle, the TNC displays the **WRITE LOG TO FILE** soft key. If you press this soft key, the TNC will record the current values determined in the active touch probe cycle.

If you store the measuring results, the TNC creates the text file TCHPRMAN.TXT. If you have not defined a path in the machine parameter **fn16DefaultPath**(no. 102202), the TNC will store the TCHPRMAN.TXT and TCHPRMAN.html files in the main directory **TNC:**.



Format and content of the TCHPRMAN.TXT file are preset by the machine tool builder.

Writing measured values from the touch probe cycles to a datum table



Use this function to save measured values in the workpiece coordinate system. If you want to save measured values in the workpiece coordinate system (REF coordinates), press the **ENTRY IN PRESET TABLE** soft key.

Further Information: "Writing measured values from the touch probe cycles to the preset table", page 368

With the **ENTER IN DATUM TABLE** soft key, the TNC can write the values measured during any touch probe cycle as applicable 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 box
- ▶ Press the **ENTER IN DATUM TABLE** soft key; the TNC saves the datum in the specified datum table under the entered number

Manual Operation and Setup

12.6 Using a 3-D touch probe (option 17)

Writing measured values from the touch probe cycles to the preset table



Use this function if you want to save measured values in the machine coordinate system (REF coordinates). If you want to save measured values in the workpiece coordinate system, press the **ENTER IN DATUM TABLE** soft key.

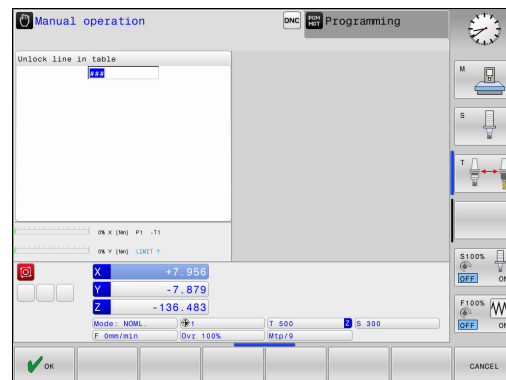
Further Information: "Writing measured values from the touch probe cycles to a datum table", page 367

With the **ENTRY IN PRESET TABLE** soft key, the TNC 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 directory TNC:\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 preset number in the **Number in table:** input box
- ▶ Press the **ENTRY IN PRESET TABLE** soft key. The TNC saves the datum in the preset table under the entered number
 - Preset number is not available: The TNC saves the row only after pressing the **OK** soft key (Create row in table?)
 - Preset number is protected: Press the **OK** soft key and the active preset will be overwritten
 - Preset number is password-protected: Press the **OK** soft key, enter the password and the active preset will be overwritten



If writing to the table row is not possible due to a lock, the control displays a message. The probing is not aborted, however.



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 calibrate the touch probe, otherwise the TNC cannot provide precise measuring results.



Always calibrate a touch probe in the following cases:

- Commissioning
- 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 TNC 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 TNC 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 of the TNC

Soft key	Function	Page
	Calibrating the length	370
	Measure the radius and the center offset using a calibration ring	371
	Measure the radius and the center offset using a stud or a calibration pin	371
	Measure the radius and the center offset using a calibration sphere	371

12.7 Calibrating 3-D touch probes (option 17)

Calibrating the effective length

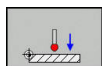


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

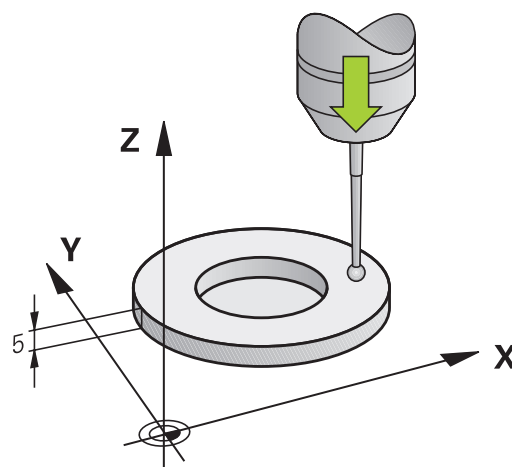


The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.

- ▶ Set the datum 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 TNC 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 TNC logs the calibration process in TCHPRMAN.html.



Calibrating the effective radius and compensating center misalignment

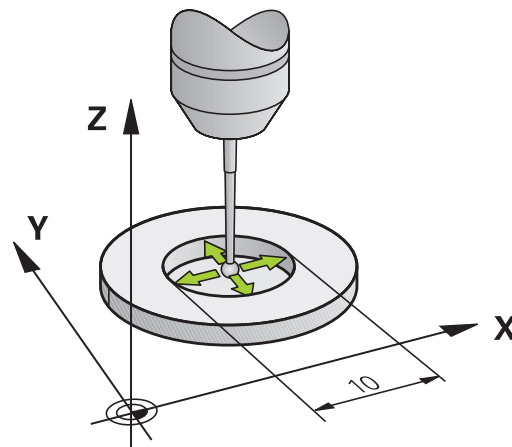


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



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.



When calibrating the ball tip radius, the TNC executes an automatic probing routine. During the first cycle, the TNC determines the center of the calibration ring or stud (rough measurement) and positions the touch probe in the center. Then the ball tip radius is determined during the actual calibration process (fine measurement). If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

The characteristic of whether and how your touch probe can be oriented is predefined in HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

The calibration routine varies depending on how your touch probe can be oriented:

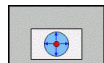
- No orientation possible or orientation possible in only one direction: The TNC executes one rough and one fine measurement and determines the effective ball tip radius (column R in tool.t)
- Orientation possible in two directions (e.g. HEIDENHAIN wired touch probes): The TNC executes one rough and one fine measurement, rotates the touch probe by 180° and then completes one more probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations
- Orientation possible in any direction (e.g. HEIDENHAIN infrared systems): The TNC executes one rough and one fine measurement, rotates the touch probe by 180° and then completes one more probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations

Manual Operation and Setup

12.7 Calibrating 3-D touch probes (option 17)

Calibration using a calibration ring

Proceed as follows for manual calibration using a calibration ring:



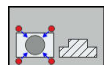
- ▶ In the **Manual operation** mode, position the ball tip inside the bore of the ring gauge
- ▶ Select the calibration function: Press the **CAL. R** soft key
- ▶ The TNC displays the current calibration data.
- ▶ Enter the diameter of the ring gauge
- ▶ Probe: Press the **NC START** key
- ▶ The 3-D 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 TNC 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 TNC logs the calibration process in TCHPRMAN.html.



In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. Refer to your machine manual.

Calibration with a stud or calibration pin

Proceed as follows for manual calibration with a stud or calibration pin:



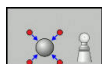
- ▶ In the **Manual operation** mode, position the ball tip above the center of the calibration pin
- ▶ Select the calibration function: Press the **CAL. R** soft key
- ▶ Enter the outside diameter of the stud
- ▶ Enter the safety clearance
- ▶ Probe: Press the **NC START** key
- ▶ The 3-D touch probe 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 TNC 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 TNC logs the calibration process in TCHPRMAN.html.



In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer.
Refer to your machine manual.

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 TNC 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 TNC logs the calibration process in TCHPRMAN.html.



In order to be able to determine the ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer.
Refer to your machine manual.

12


Manual Operation and Setup

12.7 Calibrating 3-D touch probes (option 17)

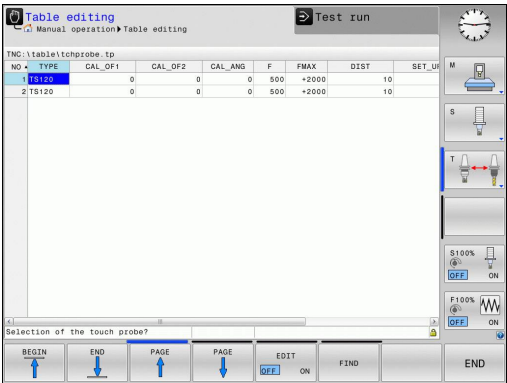
Displaying calibration values

The TNC saves the effective length and effective radius of the touch probe in the tool table. The TNC 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 TNC automatically creates the TCHPRMAN.html log file to which the calibration values are saved.






Please make sure the correct tool number is active when you use the touch probe system.



12.8 Datum setting with a 3-D touch probe (option number 17)

Overview

The following soft-key functions are available for setting a datum on an aligned workpiece:

Soft key	Function	Page
	Datum setting on any axis with	375
	Setting a circle center as datum	376
	Center line as datum Setting the center line as datum	379

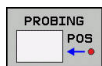


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

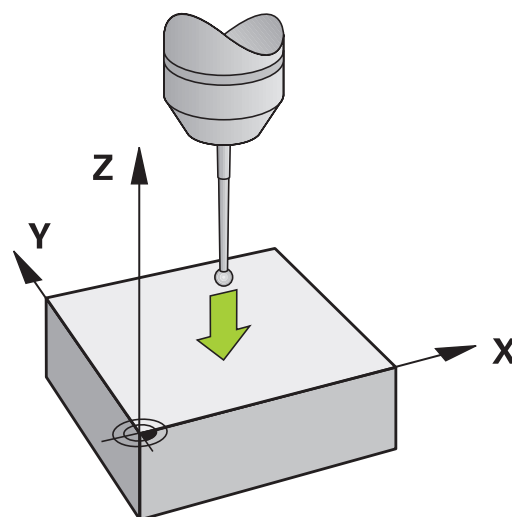


Please ensure that the TNC refers to the probed value on the active datum or the last defined datum in **MANUAL OPERATION** mode with active datum shift. The datum shift is included in the position display.

Datum setting 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
- ▶ **Datum:** Enter the nominal coordinates, confirm with the **SET DATUM** soft key
- ▶ **Further Information:** "Writing measured values from the touch probe cycles to a datum table", page 367
- ▶ To terminate the probe function, press the **END** soft key



12.8 Datum setting with a 3-D touch probe (option number 17)

Circle center as datum

With this function, you can set the datum at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

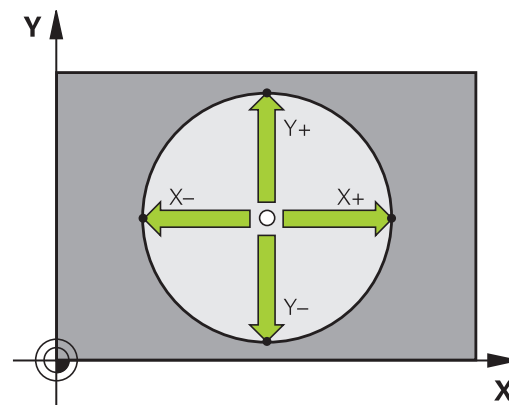
Inside circle:

The TNC 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 TNC calculate the center (four touch points are recommended)
- ▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- ▶ **Datum:** In the menu window, enter both coordinates of the circle center, confirm with the **SET DATUM** soft key, or write the values to a table
Further Information: "Writing measured values from the touch probe cycles to a datum table", page 367
Further Information: "Writing measured values from the touch probe cycles to the preset table", page 368
- ▶ To terminate the probe function, press the **END** soft key



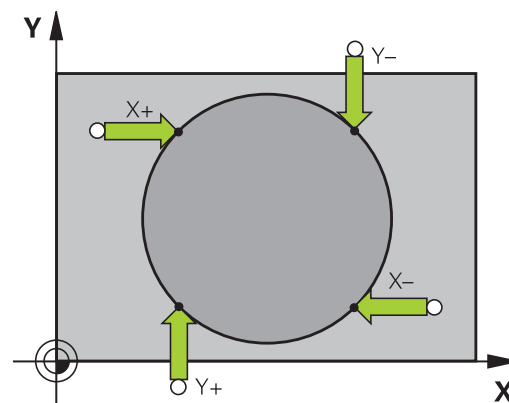
The TNC needs only three touch points to calculate outside or inside circles, e.g. with circle segments. More precise results are obtained if you measure circles using four touch points, however. You should always preposition the touch probe in the center, or as close to the center as possible.

Datum setting with a 3-D touch probe (option number 17) 12.8

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 TNC calculate the center (four touch points are recommended)
- ▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- ▶ **Datum:** Enter the coordinates of the datum, confirm with the **SET DATUM** soft key, or write the values to a table
Further Information: "Writing measured values from the touch probe cycles to a datum table", page 367
Further Information: "Writing measured values from the touch probe cycles to the preset table", page 368)
- ▶ To terminate the probe function, press the **END** soft key



Once the probing routine is completed, the TNC displays the current coordinates of the circle center and the circle radius.

Manual Operation and Setup

12.8 Datum setting with a 3-D touch probe (option number 17)

Setting the datum using multiple holes/cylindrical studs

The manual probing function **Model Circle** 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 (model circle)** for using multiple holes or circular studs to set the datum. You can set the intersection of two or more elements as datum.

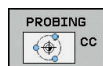
Setting the datum in the intersection of multiple holes/circular studs:

- Pre-position touch probe

Select **Model Circle** probing function

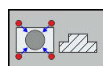


- Select the touch probe function: Press the **PROBING CC** soft key

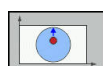


- Press the **PROBING CC (model circle)** soft key

Probe a circular stud



- Circular stud should be probed automatically: Press **STUD** soft key



- Enter or select using soft key

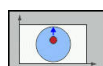


- Start probing function: Press the **NC START** soft key

Probe the hole.



- Hole should be probed automatically: Press the **HOLE** soft key



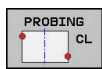
- Enter or select using soft key



- Start probing function: Press the **NC START** soft key
- Repeat the probing procedure for the remaining elements
- Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key
- **Datum:** In the menu window, enter both coordinates of the circle center, confirm with the **SET DATUM** soft key, or write the values to a table
Further Information: "Writing measured values from the touch probe cycles to a datum table", page 367
Further Information: "Writing measured values from the touch probe cycles to the preset table", page 368
- To terminate the probe function, press the **END** soft key

Datum setting with a 3-D touch probe (option number 17) 12.8

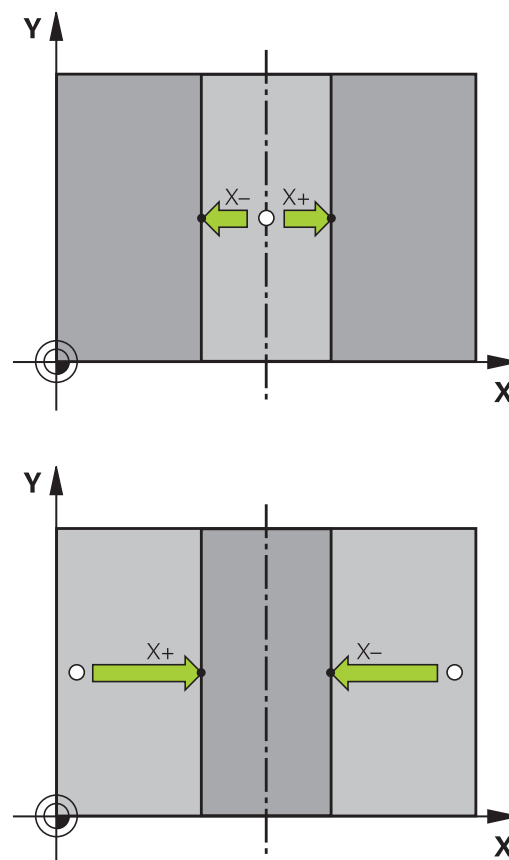
Setting a center line as datum



- ▶ 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
- ▶ **Datum:** Enter the coordinates of the datum in the menu window, confirm with the **SET DATUM** soft key, or write the value to a table
Further Information: "Writing measured values from the touch probe cycles to a datum table", page 367
Further Information: "Writing measured values from the touch probe cycles to the preset table", page 368
- ▶ To terminate the probe function, press the **END** soft key



After you have measured the second touch point, you can use the evaluation menu to change the direction of the centerline. Using the soft keys, you can choose whether the datum should be set in the principal axis, minor axis or tool axis. This may be required if you want to save the set position on the principal or minor axis.



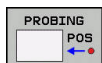
12.8 Datum setting with a 3-D touch probe (option number 17)

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

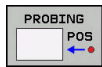
Finding the coordinates of a position on an aligned workpiece

- ▶ 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

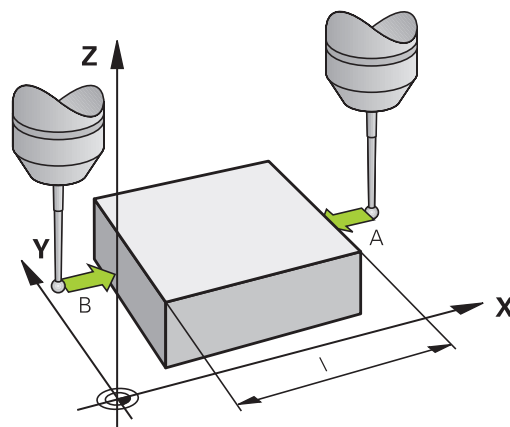
The TNC shows the coordinates of the touch point as reference point.

Datum setting with a 3-D touch probe (option number 17) 12.8

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 datum later, write down the value that appears in the Datum display
- ▶ Datum: 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 datum that was active before the length measurement:

- ▶ Select the probing function: Press the **PROBING POS** soft key
- ▶ Probe the first touch point again
- ▶ Set the datum to the value that you wrote down previously
- ▶ Cancel the dialog: Press the **END** key

13

**Positioning with
Manual Data Input**

13

Positioning with Manual Data Input

13.1 Programming and executing simple machining operations

13.1 Programming and executing simple machining operations

The **Positioning with manl.data input** mode of operation is particularly convenient for simple machining operations or to pre-position the tool. in HEIDENHAIN Klartext conversational programming 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

In the **Positioning with manl.data input** mode of operation, the additional status display can also be activated.

Positioning with manual data input (MDI)



- ▶ Switch to the **Positioning with manl.data input** mode of operation



- ▶ 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 384



Limitation

The following functions are not available in the **Positioning with manl.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 242

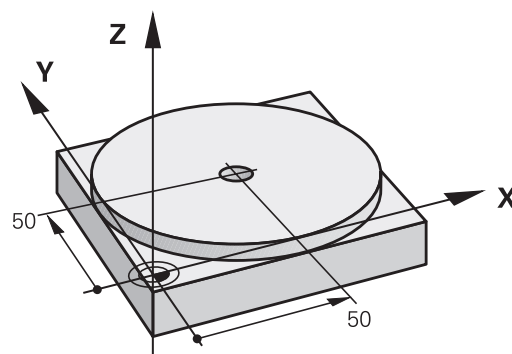
13 Positioning with Manual Data Input

13.1 Programming and executing simple machining operations

Example 1

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation 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	
Q200=5 ;SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q201=-15 ;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	
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 479

Example 2: Correcting workpiece misalignment on machines with rotary tables

- ▶ 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 operating mode: **Positioning with manl.data input**



- ▶ 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

13

Positioning with Manual Data Input

13.1 Programming and executing simple machining operations

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:



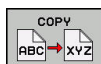
- ▶ Operating mode: Press the **Programming** key



- ▶ 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 132

14

**Test Run and
Program Run**

Test Run and Program Run

14.1 Graphics

14.1 Graphics

Application

In operating modes **Program run, single block** and **Program run, full sequence** and the operating mode **Test run** the TNC graphically simulates a machining operation.

The TNC features the following views:

- Plan view
- Projection in three planes
- 3-D view



In the **Test run** operating mode, you can also use the 3-D line graphics.

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill.

If a tool table is active, the TNC also considers the entries in the LCUTS, TANGLE and R2 columns.

The TNC 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



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 TNC 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





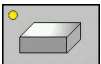
- ▶ Select the desired function by soft key, e.g. incrementally increasing the simulation speed

Test Run and Program Run

14.1 Graphics

Overview: Display modes




In operating modes **Program run, single block** and **Program run, full sequence** and in the operating mode **Test run** the TNC 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 result of the simulation can be faulty if the TNC's computer is overloaded with complicated processing tasks.

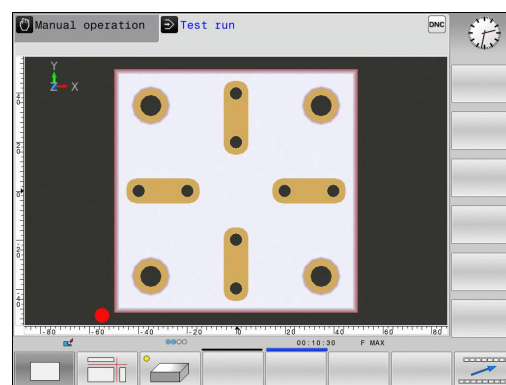
Plan view

Select the plan view in the **Test run** operating mode:

-  ▶ 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**:

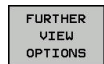
-  ▶ Press the **GRAPHICS** soft key
-  ▶ Press the **PLAN VIEW** soft key



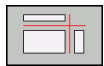
Projection in three planes

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:



- Press the **FURTHER VIEW OPTIONS** soft key

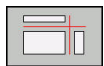


- Press the **VIEW ON 3 PLANES** soft key

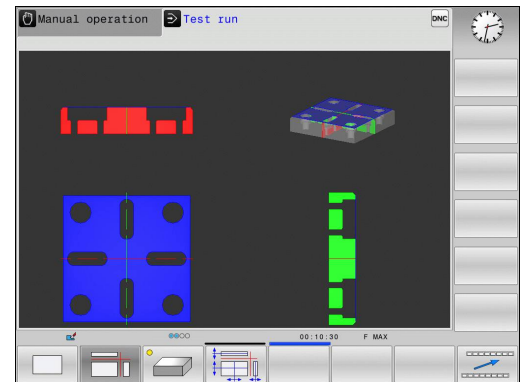
Select projection in three planes in the operating modes **Program run, single block** and **Program run, full sequence**:



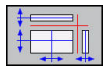
- Press the **GRAPHICS** soft key



- Press the **VIEW ON 3 PLANES** soft key



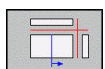
Move the sectional planes



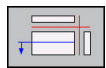
- Select the functions for shifting the sectional plane. The TNC offers the following soft keys:

Soft keys

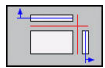
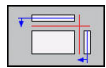
Function



Shift the vertical sectional plane to the right or left



Shift the vertical sectional plane forward or backward

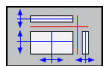


Shift the horizontal sectional plane upwards or downwards

The position of the sectional planes is visible during shifting.

The 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.

Return sectional planes to default setting:



- Select the function for resetting the sectional planes.

Test Run and Program Run

14.1 Graphics

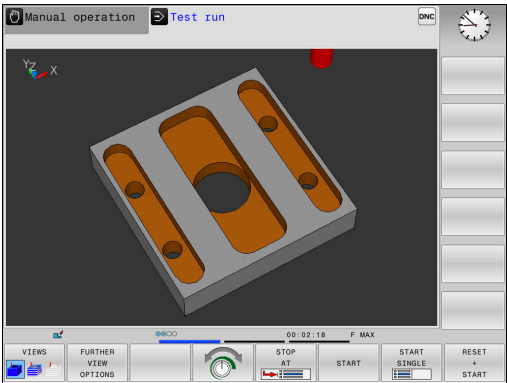
3-D view

Choose 3-D view:

The high-resolution 3-D view enables you to display the surface of the machined workpiece in greater detail. With a simulated light source, the TNC creates realistic light and shadow conditions.



- Press the 3-D view soft key



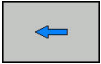
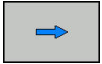



Rotating, enlarging and shifting the 3-D view



- Select the functions for rotating and enlarging: The TNC 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

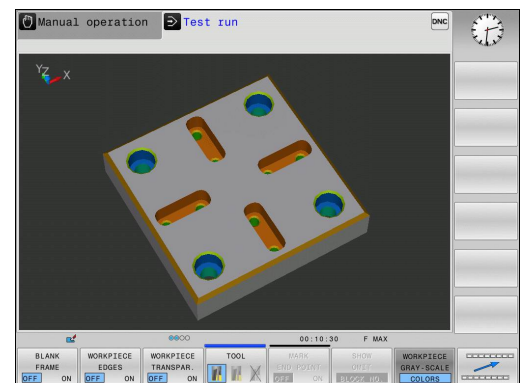
You can also use the mouse to change the graphic display. The following functions are available:

- ▶ 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 TNC 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

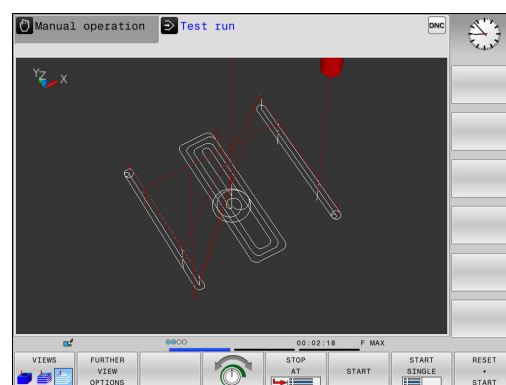
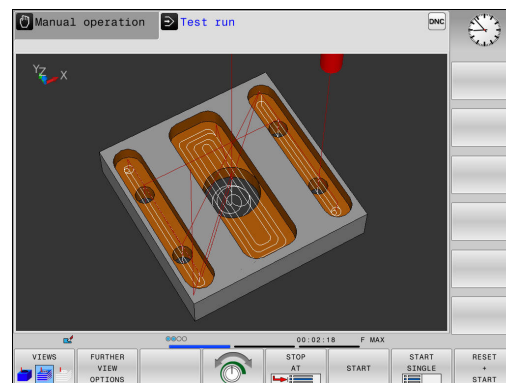


Test Run and Program Run

14.1 Graphics

The **Test run** mode of operation also offers the following functions:

Soft keys	Function
	Show workpiece blank frame
	Highlight workpiece edges on 3-D model
	Show a transparent workpiece
	Show the end points of the tool paths
	Show the block numbers of the tool paths
	Show the workpiece in color
	Reset the volume model
	Reset the tool paths
	Display the rapid traverse movements
	<p>Activate measuring</p> <p>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.</p>



Note that the range of functions depends on the model quality selected. You can select the model quality in the MOD function **Graphic settings**.





By showing the tool paths you can depict the programmed paths of the TNC in three dimensions. A powerful zoom function is available for recognizing the details quickly.

You can use the tool paths display to inspect programs created externally for irregularities before machining. This can help you to avoid undesirable machining marks on the workpiece. If points were output wrongly by the the postprocessor, machining marks may arise.

The TNC shows 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
	Display unmachined blank in the operating modes Program run, single block and Program run, full sequence
	Display unmachined blank in the operating mode Test run

Tool display

You can show the tools during the simulation.

Soft key	Function
	Program run, full sequence / Program run, single block
	Test run

Test Run and Program Run

14.1 Graphics

Measurement of machining time

Machining time in the Test Run mode of operation




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 calculated by the control can only conditionally be used for calculating the production time because the control does not account for machine-dependent times, such as tool change.

Machining time in the machine operating modes

Time display from program start to program end. The timer stops whenever machining is interrupted.

Activating the stopwatch function

-  ▶ Shift the soft-key row until the soft-key for the stopwatch functions appears
-  ▶ Select the stopwatch functions
-  ▶ 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

Showing the workpiece blank in the working space 14.2

14.2 Showing the workpiece blank in the working space

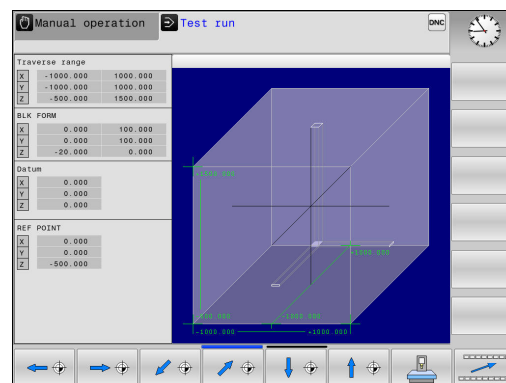
Application

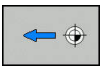
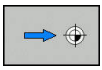




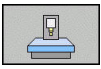
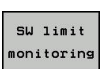
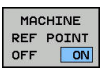
In the **Test run** operating mode, you can graphically check the position of the workpiece blank or datum in the machine's working space and activate work space monitoring in the **Test run** mode: Press the **BLANK IN WORK SPACE** soft key to activate this function. Use the soft key **SW LIMIT MONITORING** (in the second soft key row) to activate or deactivate the function.

A transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The TNC takes 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. However, if you activate working-space 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 datum 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 datum
	Switch monitoring function on or off
	Display machine reference point







Note that even with **BLK FORM CYLINDER**, a cuboid is shown in the working space as workpiece blank.

14.3 Functions for program display

Overview

In the **Program run single block** and **Program run full sequence** operating mode, the TNC displays the following soft keys for displaying a machining program in pages:

Soft key	Functions
	Go back one screen of the program
	Go forward one screen of the 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 to reduce programming errors when programs are running. The TNC checks the programs for the following:

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

The following functions are also available:

- Blockwise test run
- Interruption of test at any block
- Optional block skip
- Functions for graphic simulation
- Measure machining time
- Additional status display

14.4 Test run

**Danger of collision!**

The TNC cannot graphically simulate all traverse motions actually performed by the machine. These include

- Traverse motions during tool change, if the machine manufacturer defined them in a tool-change macro or via the PLC
- Positioning movements that the machine manufacturer defined in an M-function macro
- Positioning movements that the machine manufacturer performs via the PLC

HEIDENHAIN therefore recommends proceeding with caution for every new program, even when the program test did not output any error message, and no visible damage to the workpiece occurred.

With cuboid workpiece blanks, the TNC starts a program 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**

In order to ensure unambiguous behavior during program run, after a tool change you should always move to a position from which the TNC can position the tool for machining without causing a collision.



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. Refer to your machine manual.

Execute test run



If the central tool file is active, a tool table must be active (status S) to conduct a test run. 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 momentarily active datum from **Preset.pr** (execution). Line 0 is selected when starting the test run until you define another datum in the NC program. All datums 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 activate working space monitoring for the test run.

Further Information: "Showing the workpiece blank in the working space", page 399



- ▶ 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 TNC then displays the following soft keys:

Soft key	Functions
	Reset the blank form, reset the previous tool data and test the entire program
	Test the entire program
	Test each NC block individually
	Executes the Test run until block N
	Halt test run (soft key only appears once you have started the test run)

You can interrupt the test run and continue it again at any point—even within a fixed cycle. 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 and Program Run

14.4 Test run

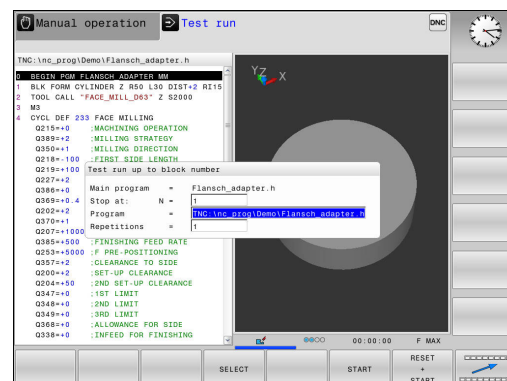
Test run up to a certain block

With the **STOP AT** function the TNC executes a **Test run** up to the block with block number **N**.

To stop the **Test run** at any block, proceed as follows:



- ▶ 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 displays the name of the selected program; if the stop should be executed in a program that was called with **PGM CALL**, 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

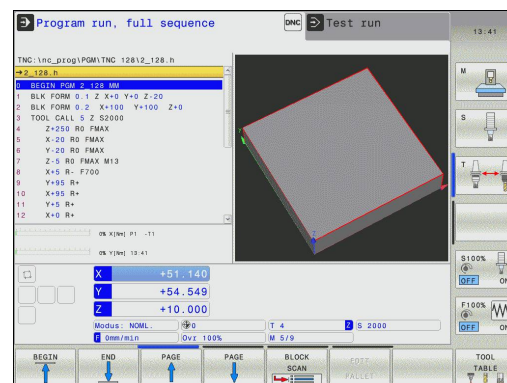
Application

In the **Program run, full sequence** operating mode, the TNC executes a machining program continuously to its end or up to a program stop.

In the **Program run, single block** operating mode, the TNC 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 TNC functions in the operating modes **Program run, single block** and **Program run, full sequence**:

- Interrupt program run
- Start 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 datum
- 3 Select the pallet files (status M)
- 4 Select the part program (status M)



You can change the feed rate and spindle speed using the potentiometers.



Refer to your machine manual. The behavior of this function varies depending on the respective machine. You can reduce the feed rate with the **FMAX** soft key. The reduction applies to all rapid traverse and feed rate movements. The value you enter remains active after switch-off or switch-on.

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

Test Run and Program Run

14.5 Program run

Interrupt, stop or abort 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 conjunction with the **INTERNAL STOP** soft key **INTERNAL STOP**
- 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

The difference between an interrupted (terminated) program run and a stopped run is that an interrupted run allows the user to carry out the following actions:

- 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 **/**



If the control registers an important error during a program run, it automatically stops the program run.
Example: Cycle call with stationary spindle

Program-controlled interruptions

You can define interruptions directly in the machining program. The control interrupts the program run in the NC Block containing one of the following inputs:

- Programmed stop **M0**
- Conditional stop **M1**



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 the **NC 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

14.5 Program run

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 reference point during an interruption

If you modify the active reference point 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



If you interrupt an NC program using the **INTERNAL STOP** key, you have to start machining at the start of the program or 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.

If you interrupt the program run within a program section repetition, or within a subprogram, you must return to the interruption point using the **BLOCK SCAN** function.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations (e.g. datum shift, mirroring)
- The coordinates of the circle center that was last defined



Note that the stored data remain active until they are reset (e.g. if you select a new program).

The TNC uses the stored data for returning the tool to the contour after manual machine axis positioning during an interruption (**RESTORE POSITION** soft key).

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

14.5 Program run

Retraction after a power interruption



The **Retraction** mode is enabled and adapted by the machine manufacturer. Refer to your machine manual.

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 with 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 TNC selects the mode of traverse and the associated parameters automatically. If the traverse mode oder the parameters have not been correctly preselected, you are unable to reset them manually.

**Danger of collision!**

The TNC uses the last stored axis values for undefined axes. These axis values may not exactly correspond with the actual axis positions!

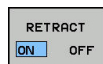
Consequently, the TNC may be unable to move the tool exactly along the actual tool direction when moving forward in the tool direction. If the tool is still in contact with the workpiece, it can cause stress or damage to the tool and workpiece. Stress or damage to the workpiece or tool can also be caused by uncontrolled coasting or braking of axes after a power interruption. If the tool is still in contact with the workpiece, move the axes carefully. Set the feed rate override to the smallest values possible. If you use the handwheel, use a small feed rate factor.

The traverse mode monitoring mode is not available for undefined axes. Watch the axes while you are moving them. Do not move to the traverse mode limits.

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 TNC starts the operating system. This process may take several minutes. Then the TNC will display the message "Power interrupted" in the screen header



- ▶ Activate the **Retraction** mode: Press the **RETRACT** soft key. The TNC displays the message **Retraction selected**



- ▶ Acknowledge the power interruption: Press the **CE** key. The TNC compiles the PLC program.



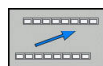
- ▶ Switch on the control voltage: The TNC checks the functioning of the EMERGENCY STOP circuit. If there is at least one undefined axis you will have to compare the displayed positions with the actual axis values and confirm they are correct, and, if needed, following any instructions given in the dialog

- ▶ 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

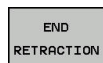


- ▶ Activate retraction: Press the **RETRACT** soft key

- ▶ 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 the **Retraction** mode: Press the **END RETRACTION** soft key. The TNC checks whether the **Retract** mode can be exited, following any instructions given in 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 TNC hides **Retraction selected** mode
- ▶ Initialize the machine: if required, cross the datums
- ▶ Establish the desired machine condition: If required, reset the tilted working plane

Any entry into program: Mid-program startup

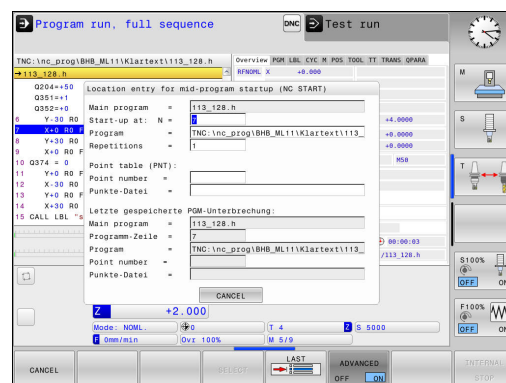
**Danger of collision!**

If you use the **GOTO** key and the number of the NC block for going into a program, neither the TNC nor the PLC will execute any functions that ensure a safe start.

For safe entry, always use the **BLOCK SCAN** function.



The **BLOCK SCAN** function must be enabled and adapted by the machine manufacturer. Refer to your machine manual.



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 **INTERNAL STOP**
- Emergency stop
- Power failure

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 change between **Program run full sequence** and **Program run single block**.

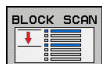


The TNC skips all touch probe cycles in a midprogramstartup. Result parameters from these cycles might therefore remain empty. If you wish to continue working with the results of a touch probe cycle, use the multi-level mid-program startup.

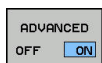
Procedure for simple mid-program startup



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 machining operations which should be taken into account in the block scan if the NC block is located within a program section repetition.
Default 1 means first machining



- ▶ Press the **ADVANCED** soft key if required 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 416



- ▶ Press the **NC START** key
- > The control resumes execution of the NC program.

Example of simple mid-program startup

After an internal stop you wish to start in block 12 in the third machining of LBL 1.

In the pop-up window enter the following data:

- **Start-up at: N =12**
- **Repetitions = 3**

Test Run and Program Run

14.5 Program run

Procedure for multi-level mid-program startup

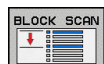
If you start in a subprogram that is called several times by the main program, use the multi-level mid-program startup. For this purpose, jump in the main program to the desired subprogram call. Use the function **CONTINUE BLOCK SCAN** to jump further from this position.



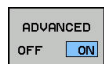
The control only displays the dialogs required by the process in the pop-up window.

You can also jump to the next startup point without restoring the machine status and the axis positions of the first startup point. For this purpose press the soft key **CONTINUE BLOCK SCAN** before you confirm 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 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:

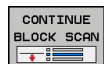


- ▶ Select the **Program run single block** operating mode if required



- ▶ 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

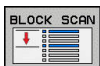
- ▶ Repeat these steps if required to jump to the next start-up point
- ▶ Press the **NC START** key
- ▶ The control resumes execution of the NC program.



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**



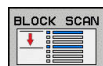
- ▶ Press the **NC START** key until the control runs the NC block
- ▶ The control continues to run the subprogram and then returns to the main program.

Test Run and Program Run

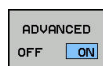
14.5 Program run

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
- ▶ **Point file** = Enter the name and path of the point table



- ▶ Press the **SELECT LAST BLOCK** soft key if required to select the last saved interruption

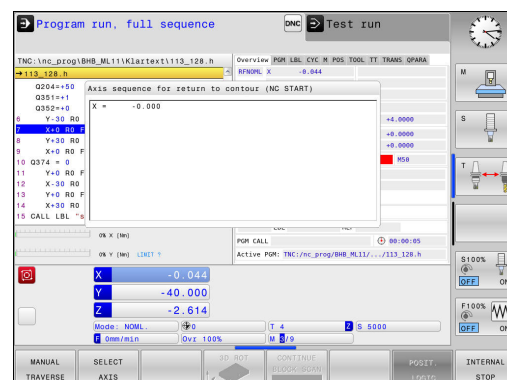


- ▶ Press the **NC START** key

Returning to the contour

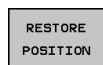
With the **RESTORE POSITION** function, the TNC moves the tool to the workpiece contour in the following situations:

- 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

To move to the contour, proceed as follows:



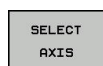
- ▶ Press the **RESTORE POSITION** soft key
- ▶ Restore the machine status, if required

Approach the axes in the sequence shown by the control:



- ▶ Press the **NC START** key

Approach the axes according to individually selected sequence:



- ▶ 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

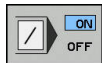


- ▶ Repeat the process for all axes

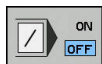
14.6 Optional block skip

Application

Blocks that you mark with a "/" sign may be skipped in **Test run** or **Program Run, Full Sequence/Single Block**:



- ▶ To run or test the program without the NC blocks preceded by a slash, set the soft key to **ON**



- ▶ To run or test the program with the NC blocks preceded by a slash, set the soft key to **OFF**



This function does not work for **TOOL DEF** blocks.
After a power interruption the TNC returns to the most recently selected setting.

Inserting the "/" character

- ▶ In the **Programming** mode you select the block in which the character is to be added



- ▶ Press the **INSERT** soft key

Erasing the "/" character

- ▶ In the **Programming** mode you select the block in which the character is to be erased



- ▶ Press the **REMOVE** soft key

Test Run and Program Run

14.7 Optional program-run interruption

14.7 Optional program-run interruption

Application



Refer to your machine manual.

The behavior of this function varies depending on the respective machine.

The TNC optionally interrupts program run at blocks containing M1. If you use M1 in the **Program run** mode, the TNC does not switch off the spindle or 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

MOD Functions

15.1 MOD function

15.1 MOD function

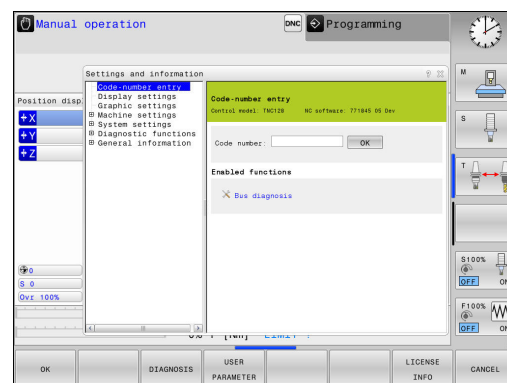
The MOD functions provide additional input possibilities and displays. In addition, you can enter code numbers to enable access to protected areas.

Selecting MOD functions

Open the pop-up window with the MOD functions:



- ▶ To select the MOD functions, press the **MOD** key. The TNC 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 a setting by pressing the **ENT** key, e.g. when setting program input
- ▶ Change a setting via a selection window



If more than one possibility is available for a particular setting, you can superimpose a window listing all of the given possibilities by pressing the **GOTO** key. Select the 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

Machine settings

- Kinematics
- Traverse limits
- Tool-usage file
- External access

System settings

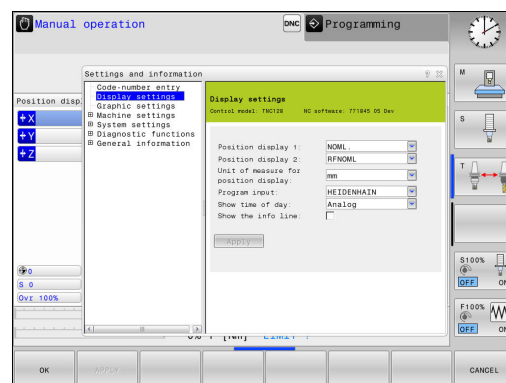
- Set the system time
- Define the network connection
- Network: IP configuration

Diagnostic functions

- Bus diagnosis
- HEROS information

General information

- Software version
- FCL information
- License information
- Machine times



MOD Functions

15.2 Graphic settings

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:

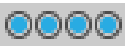
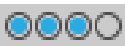
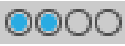

- ▶ Select the group **Graphic settings** from the MOD menu
- ▶ Select the model type
- ▶ Select the model quality
- ▶ Press the **APPLY** soft key
- ▶ Press the **OK** soft key.

You have the following simulation parameters for the graphic settings:

Model type

Displayed symbol	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

Displayed symbol	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

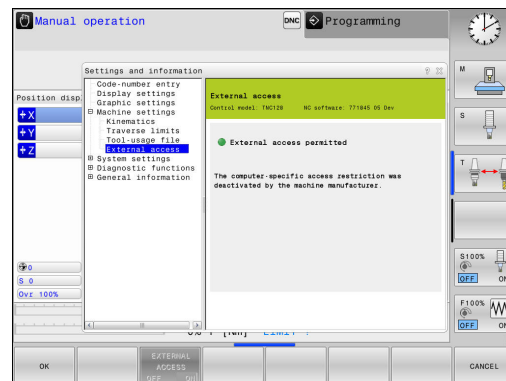


Refer to your machine manual. The machine tool builder can configure the external access options.

With the MOD function **External access** you can grant or restrict access to the TNC. If you have restricted the external access it is no longer possible to connect to the TNC and exchange data via a network or a serial connection, e.g. with the TNCremo data transfer software.

Restricting external access:

- ▶ In the MOD menu select the **Machine settings** group **Machine settings**
- ▶ Select the **External access** menu
- ▶ Set the **EXTERNAL ACCESS ON/OFF** soft key to **OFF**
- ▶ Press the **OK** soft key



15.3 Machine settings

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 TNC opens an input window 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:

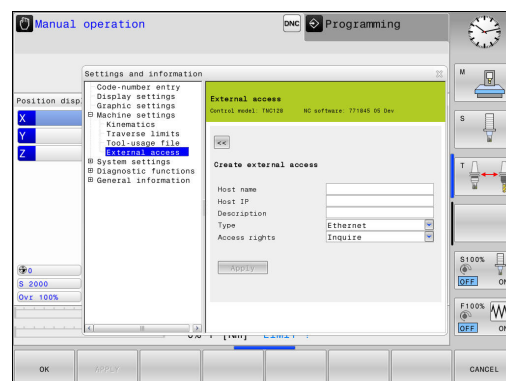
Inquire	The TNC opens a query dialog with external access
Deny	Do not permit network access
Permit	Permit network access without query

If you assign the access right **Inquire** to a connection and access is implemented from this address, the TNC 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.

The **Traverse limits** function must be enabled and adapted by the machine manufacturer.

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 to protect a component from collision for example.

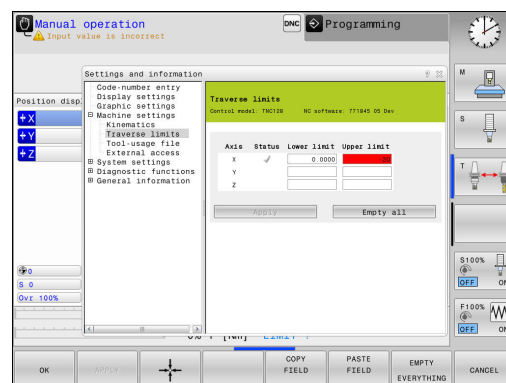
To enter traverse limits:

- ▶ In the MOD menu select the **Machine settings** group **Machine settings**
- ▶ Select the **Traverse limits** menu **Traverse limits**
- ▶ 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 TNC checks the values entered for validity.
- ▶ Press the soft key **OK**



The protection zone becomes active automatically as soon as you set a valid limit in an axis. Settings are kept even after restarting the control.

You can only deactivate the protection zone by deleting all values or pressing the **EMPTY EVERYTHING** soft key.



MOD Functions

15.3 Machine settings

Tool usage file



The tool usage test function must be enabled by your machine manufacturer. Refer to your machine manual.

With the MOD function **Tool-usage file** you can select whether the TNC never, once, or always creates a tool usage file.

Generate a tool usage file:

- ▶ In the MOD menu select the **Machine settings** group **Machine settings**
- ▶ Select the **Tool-usage file** menu **Tool-usage file**
- ▶ Select the desired setting for the operating modes **Program Run, Full Sequence/Single Block** and **Test run**
- ▶ Press the **APPLY** soft key **APPLY**
- ▶ Press the **OK** soft key

Select kinematics



The **Kinematics selection** function must be enabled and adapted by the machine manufacturer. Refer to your machine manual.

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.



Danger of collision!

When you switch the kinematics model for machine operation, the TNC implements all of subsequent movements with modified kinematics.

Ensure that you have selected the correct kinematics in the test run 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.

To set the system time manually:

- ▶ In the MOD menu select the **System settings** group **System settings**
- ▶ Press the **SET DATE/ TIME** soft key **SET DATE/ TIME**
- ▶ Select your time zone in the **Time zone** area **Time zone**
- ▶ Press the **LOCAL/NTP** soft key in order to select the **Set the time manually** entry
- ▶ If required, change the datum and the time
- ▶ 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 **System settings**
- ▶ Press the **SET DATE/ TIME** soft key **SET DATE/ TIME**
- ▶ Select your time zone in the **Time zone** area **Time zone**
- ▶ Press the **LOCAL/NTP** soft key in order to synchronize the time entry through the NTP server
- ▶ Enter the host name or the URL of an NTP server
- ▶ Press the **ADD** soft key
- ▶ Press the **OK** soft key

MOD Functions

15.5 Select the position display

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:

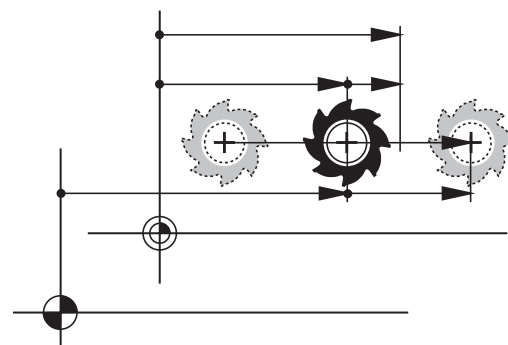
- Initial position
- Target position of the tool
- Workpiece zero point
- Machine datum

The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the value presently commanded by the TNC	NOML.
Actual position; current tool position	ACTL.
Reference position; actual position relative to the machine datum	REF ACTL
Reference position; nominal position relative to the machine datum	REF NOML
Servo lag; difference between nominal and actual positions	LAG
Distance remaining to the programmed position in the input system; difference between actual and target positions	ACTDST
Distance remaining to the programmed position with reference to the machine datum; difference between reference and target positions	REFDST
Traverses that were carried out with handwheel superimpositioning (M118)	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

This MOD function determines whether the coordinates are displayed in millimeters (metric system) 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 TNC 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

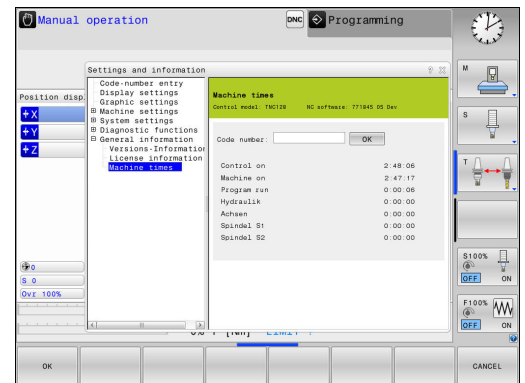
Application

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.



MOD Functions

15.8 Software numbers

15.8 Software numbers

Application

The following software numbers are displayed on the TNC 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 TNC 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 9

15.9 Enter the code number

Application

The TNC 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

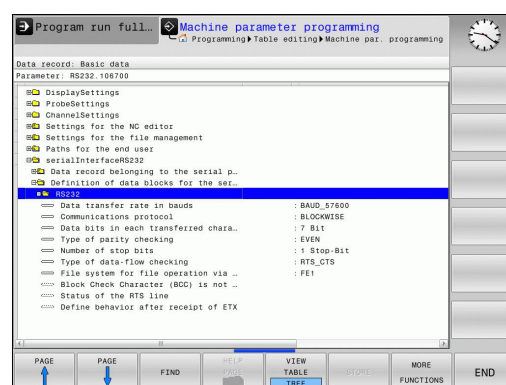
15.10 Setting up data interfaces

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 TNC 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.

MOD Functions

15.10 Setting up data interfaces

Set protocol
(protocol no. 106702)

The data transfer protocol controls the data flow of a serial transmission (comparable to MP5030 of the iTNC 530).



The BLOCKWISE setting here designates a method of data transfer in which data is transferred grouped in blocks. This should not be confused with the block-wise data reception and simultaneous block-wise running of older TNC contouring controls. The block-wise reception and simultaneous running of the same NC program is not supported by the control.

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. Corresponds to the EXT1 and EXT2 modes of earlier TNC 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 Condition of RTS line (optional) you determine whether the "low" level is active in idle state.

- TRUE: Level is "low" in idle state
- FALSE: Level is not "low" in idle state

MOD Functions

15.10 Setting up data interfaces

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)



In the FE2 and FEX operating modes you cannot use the "load all programs", "load offered program" and "load directory" functions.

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 transmitting files to and from the TNC, we recommend using the HEIDENHAIN TNCremo data transmission software. With TNCremo, data transmission is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.



You can download the current version of TNCremo free of charge from the HEIDENHAIN Filebase (www.heidenhain.com, <Software>, <PC software>, <TNCremo>).

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, TNCremo automatically tries to set up a connection with the TNC.

15.10 Setting up data interfaces

Data transfer between the TNC and TNCremo



Before you transmit a program from the TNC to the PC, you must make absolutely sure that you have already saved the program currently selected on the TNC. The TNC saves changes automatically when you switch the operating mode on the TNC, or when you select the file manager with the **PGM MGT** key.

Check whether the TNC 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 TNC and displays this at the bottom left of the main window **2**
- ▶ To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**
- ▶ To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**

If you want to control data transfer from the TNC, establish the connection with your PC in the following manner:

- ▶ Select <Extras>, <TNCserver>. TNCremo is now started in server mode, and can receive data from the TNC and send data to the TNC
 - ▶ You can now call the file management functions on the TNC by pressing the key **PGM MGT** in order to transfer the desired files
- Further Information:** "Data transfer to or from an external data carrier", page 146



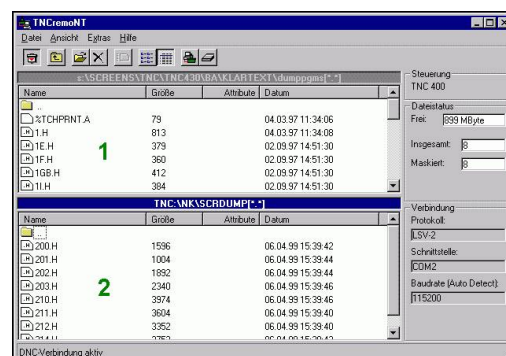
If you have exported a tool table from the control, the tool types are converted to a tool type number.

End TNCremo

Select <File>, <Exit>



Refer also to the TNCremo context-sensitive help texts where all of the functions are explained in more detail. The help texts must be called with the **F1** key.



15.11 Ethernet interface

Introduction

The TNC is shipped with a standard Ethernet card to connect the control as a client in your network. The TNC transmits data via the Ethernet card with

- the **smb** protocol (Server Message Block) for Windows operating systems, or
- the **TCP/IP** protocol family (Transmission Control Protocol/Internet Protocol) and with the NFS (Network File System)

Connection possibility

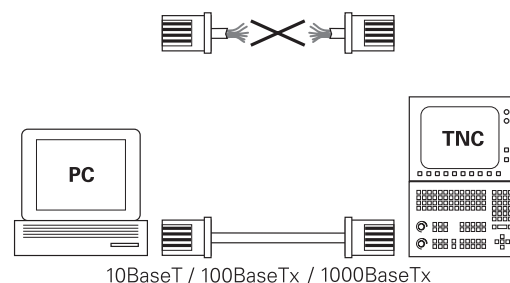
You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 1000BaseTX, 100BaseTX and 10BaseT), or directly to a PC. The connection is metallically isolated from the control electronics.

For a 1000Base TX, 100BaseTX or 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.



The maximum cable length between TNC and a node depends on the quality grade of the cable, the sheathing and the type of network (1000BaseTX, 100BaseTX or 10BaseT).

No great effort is required to connect the TNC directly to a PC that has an Ethernet card. Simply connect the TNC (port X26) and the PC with an Ethernet crossover cable (trade names: crossed patch cable or crossed STP cable).



Configuring the TNC



Make sure that the person configuring your TNC is a network specialist.

- ▶ Press the **MOD** key in the **Programming** mode and enter the code number NET123
- ▶ In the file manager, press the soft key **NET**

MOD Functions

15.11 Ethernet interface

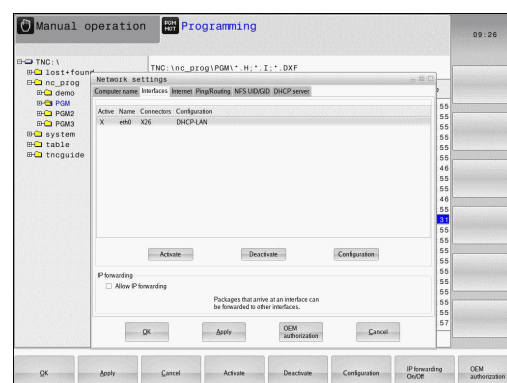
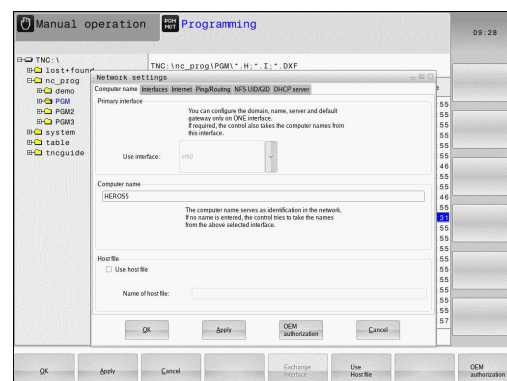
General network settings

- 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 TNC 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) <ul style="list-style-type: none"> ■ 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 external access via the second, optional Ethernet interface of the TNC is necessary for diagnostic purposes. Only do so after instruction by our Service Department



- Press the **Configuration** button to open the Configuration menu:

Setting	Meaning
Status	<ul style="list-style-type: none"> ■ 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	<p>Here you can create or select a profile in which all settings shown in this window are stored. HEIDENHAIN provides two standard profiles:</p> <ul style="list-style-type: none"> ■ DHCP-LAN: Settings for the standard TNC Ethernet interface, should work in a standard company network ■ MachineNet: Settings for the second, optional Ethernet interface; for configuration of the machine network <p>Press the corresponding buttons to save, load and delete profiles</p>
IP address	<ul style="list-style-type: none"> ■ Automatically procure IP address: The TNC 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)	<ul style="list-style-type: none"> ■ Automatically procure DNS: The TNC 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	<ul style="list-style-type: none"> ■ Automatically procure default gateway: Die TNC soll den Default-Gateway automatisch beziehen ■ 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

MOD Functions

15.11 Ethernet interface

- ▶ The **Internet** tab currently has no function.

Setting	Meaning
Proxy	<ul style="list-style-type: none"> ■ 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

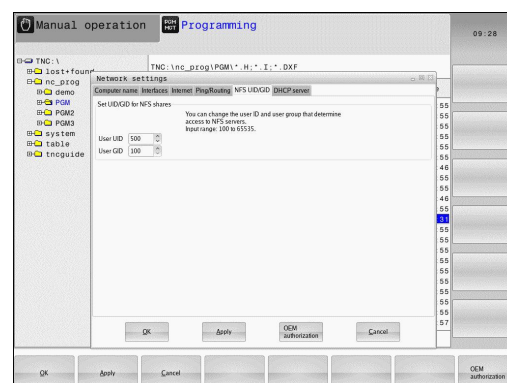
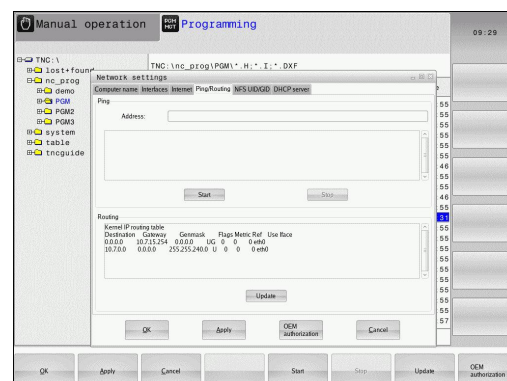
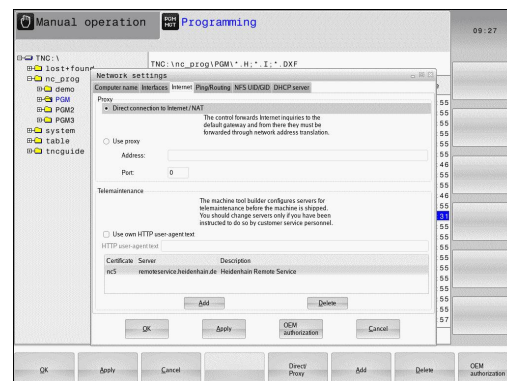
Telemaintenance The machine manufacturer configures the server for telemaintenance here. Changes must always be made in agreement with your machine tool builder

- ▶ Select the **Ping/Routing** tab to enter the ping and routing settings:

Setting	Meaning
Ping	<p>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</p> <ul style="list-style-type: none"> ■ Press the Start button to begin the test. The TNC shows the status information in the Ping field ■ Press the Stop button to conclude the test
Routing	<p>For network specialists: Status information of the operating system for the current routing</p> <ul style="list-style-type: none"> ■ Press the Update button to refresh the routing information

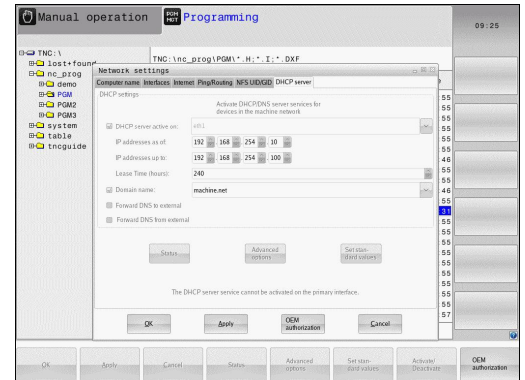
- ▶ Select the **NFS UID/GID** tab to enter the user and group identifications:

Setting	Meaning
Set UID/GID for NFS shares	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ IP addresses from: Define the IP address as of which the TNC is to derive the pool of dynamic IP addresses. The TNC 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 TNC 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 TNC reassigns the same dynamic IP address. ■ Domain name: Here you can define a name for the machine network if required. This is necessary if the same 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 TNC 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 standard values button: Set factory settings.
► Sandbox:	Changes must always be made in agreement with your machine tool builder

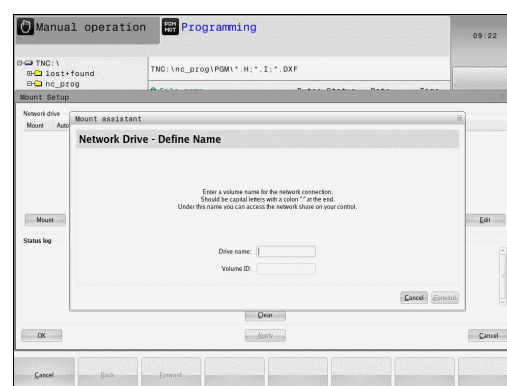
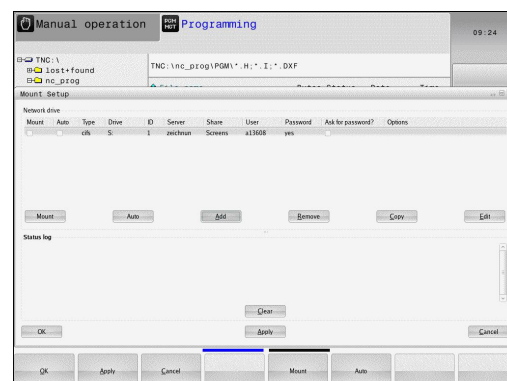


15.11 Ethernet interface

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	<p>List of all connected network drives. The TNC shows the respective status of the network connections in the columns:</p> <ul style="list-style-type: none"> ■ 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 TNC ■ 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 TNC 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 <p>To manage the network drives, use the screen buttons.</p> <p>To add network drives, use the Add button: The TNC then starts the connection wizard, which guides you by dialog through the required definitions.</p>
Status log	<p>Display of status information and error messages.</p> <p>Press the Clear button to delete the contents of the Status Log window.</p>




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 configured 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 the standard settings checked by your network specialist and change them if necessary.
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 93
- ▶ 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.

15.12 Firewall

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	<p>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</p> <ul style="list-style-type: none"> ■ 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, the 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 package for this service has been blocked. A "blue" message is output if a network packet for this service has been 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

MOD Functions

15.13 Configuring the HR 550FS wireless handwheel

15.13 Configuring the HR 550FS wireless handwheel

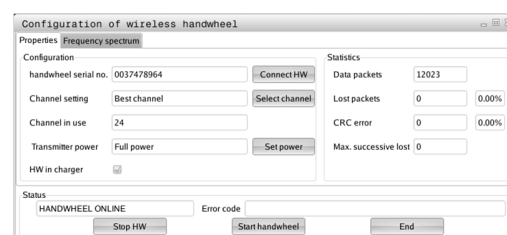
Application

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 the **Connect HW** button: The TNC 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 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 TNC 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

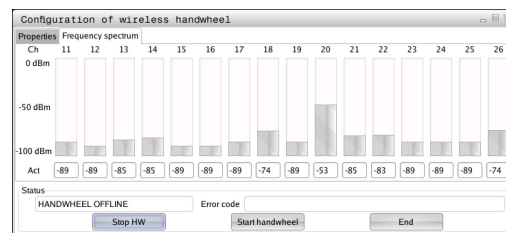
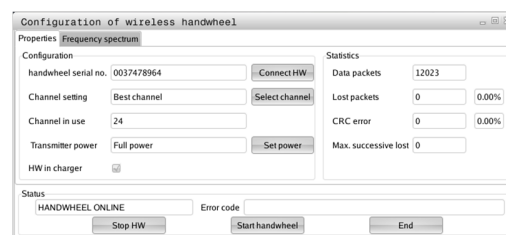


Configuring the HR 550FS wireless handwheel 15.13

Setting the transmission channel

If the wireless handwheel is started automatically, the TNC tries to select the transmission channel supplying the best transmission signal. If you want to set the transmission channel manually, 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
- ▶ Click the **Frequency spectrum** tab
- ▶ Click the **Stop HW** button: The TNC stops the connection to the wireless handwheel and determines the current frequency spectrum for all of the 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 the **Select channel** button: The TNC shows all available channel numbers. Click the channel number for which the TNC determined the least amount of radio traffic
- ▶ To save the configuration and exit the configuration menu, press the **END** button

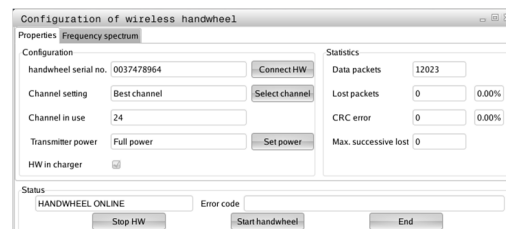


Selecting the transmitter power



Please keep in mind that the transmission range of the wireless handwheel decreases when the transmitter power is reduced.

- ▶ 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 **Set power** button: The TNC shows the three available power settings. Click the desired setting
- ▶ To save the configuration and exit the configuration menu, press the **END** button



MOD Functions

15.13 Configuring the HR 550FS wireless handwheel

Statistical data

To display the statistical data, proceed as follows:

- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ To select the configuration menu for the wireless handwheel, press the **SET UP WIRELESS HANDWHEEL** soft key: The TNC displays the configuration menu with the statistical data

Under **Statistics**, the TNC 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 TNC repeatedly displays values greater than 2 during normal operation of the wireless handwheel within the desired range of use, then there is a 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 447

Further Information: "Selecting the transmitter power", page 447



15.14 Load machine configuration

Application



Caution: Data loss!

The TNC overwrites your machine configuration when you load (restore) a backup. The overwritten machine data will be lost in the process. You can no longer undo this process!

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:

- ▶ In the MOD dialog, enter the keyword **RESTORE**
- ▶ In the TNC's file manager, select the backup file (e.g. BKUP-2013-12-12_.zip). The TNC opens a 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

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.



Danger of collision!

Cycles sometimes execute extensive operations. For safety reasons, you should run a graphical 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

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

Soft key	Cycle group	Page
	Cycles for pecking, reaming, boring, tapping and counterboring	476
	Cycles for milling rectangular pockets and rectangular studs	512
	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	536
	Cycles for producing point patterns	468
	Special cycles: dwell time, program call, oriented spindle stop	552

-  ► 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

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 456

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



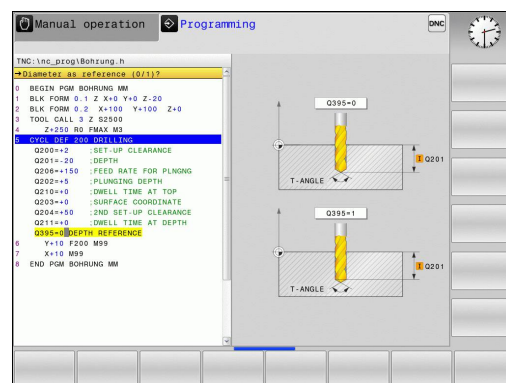
- ▶ The soft-key row shows the available groups of cycles



- ▶ 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



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



- ▶ 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

16.3 Working with fixed cycles

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.



- ▶ 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 462

Further Information: "Point tables", page 472

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

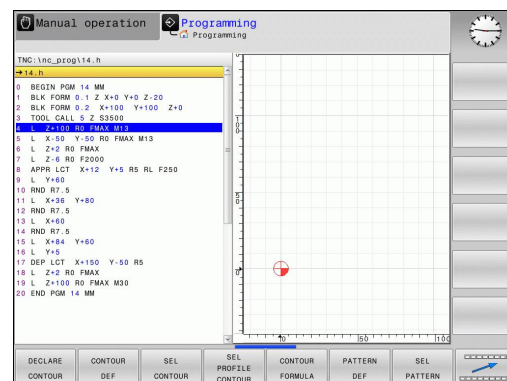
16.4 Program defaults for cycles

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.

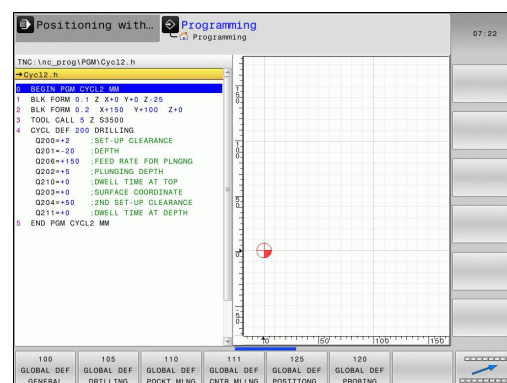
The following GLOBAL DEF functions are available:

Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	459
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	460
110 GLOBAL DEF POCKET MILLING	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	460
111 GLOBAL DEF CNTR MILLING	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	460
125 GLOBAL DEF POSITIONING	GLOBAL DEF POSITIONING Definition of the positioning behavior for CYCL CALL PAT	461
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	461



Entering GLOBAL DEF






- ▶ Operating mode: Press the **Programming** key
- ▶ Press the **SPEC FCT** key to select the special functions
- ▶ Select the functions for program defaults
- ▶ Press the **GLOBAL DEF** soft key
- ▶ 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

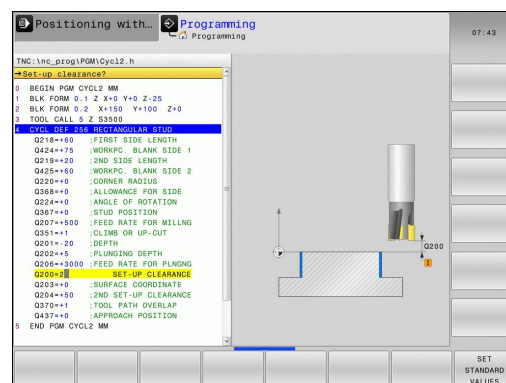


Using GLOBAL DEF information

If you have entered the corresponding GLOBAL DEF functions at the beginning of the program, then you can link to these globally valid values when defining any fixed cycle.

Proceed as follows:

-  ▶ Operating mode: Press the Programming key
-  ▶ Select machining cycles: Press the CYCLE DEF key
-  ▶ Select the desired group of cycles, for example: drilling cycles
-  ▶ Select the desired cycle, e.g. **DRILLING**
- ▶ The TNC displays the **SET STANDARD VALUES** soft key, if there is a global parameter for it
-  ▶ Press the **SET STANDARD VALUES** soft key. The TNC enters the word **PREDEF** (predefined) 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



Danger of collision!

Please note that later changes to the program settings affect the entire machining program, and can therefore change the machining procedure significantly.

If you enter a fixed value in a fixed cycle, then this value will not be changed by the **GLOBAL DEF** functions.

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.

16.4 Program defaults for cycles

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 with numbers greater than 4xx.

16.5 PATTERN DEF pattern definition

16.5 PATTERN DEF pattern definition

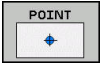

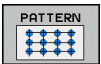
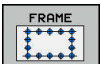


Application

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.








PATTERN DEF is to be used 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	463
	ROW Definition of a single row, straight or rotated	464
	PATTERN Definition of a single pattern, straight, rotated or distorted	465
	FRAME Definition of a single frame, straight, rotated or distorted	466
	CIRCLE Definition of a full circle	467
	PITCH CIRCLE Definition of a pitch circle	467

Entering PATTERN DEF

-  ► Operating mode: 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 456

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.

You can use the mid-program startup function to select any point at which you want to start or continue machining.

Further Information: "Any entry into program: Mid-program startup", page 412

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.

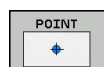
Defining individual machining positions



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.

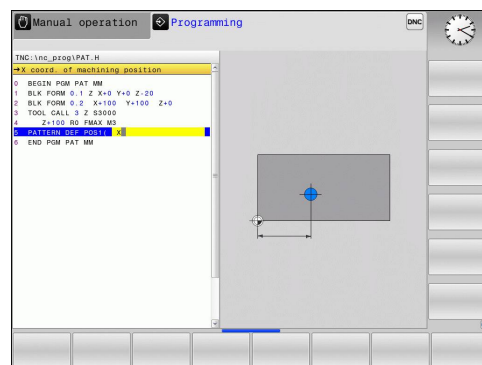


- ▶ POS1: **X coord. of machining position** (absolute): Enter the X coordinate
- ▶ POS1: **Y coord. of machining position** (absolute): Enter the Y coordinate
- ▶ POS1: **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin
- ▶ POS2: **X coord. of machining position** (absolute or incremental): Enter the X coordinate
- ▶ POS2: **Y coord. of machining position** (absolute or incremental): Enter Y coordinate
- ▶ POS2: **Z coord. of machining position** (absolute or incremental): Enter Z coordinate

NC blocks

10 Z+100 R0 FMAX

11 PATTERN DEF
POS1 (X+25 Y+33.5 Z+0)
POS2 (X+15 Y+6.5 Z+0)

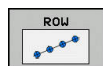


16.5 PATTERN DEF pattern definition

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.

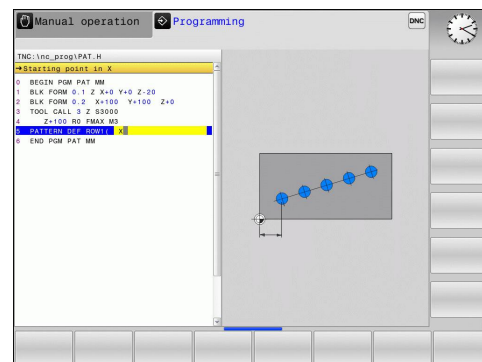


- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the row in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the row in the Y axis
- ▶ **Spacing of machining positions** (incremental): Distance between the machining positions. You can enter 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

NC blocks

10 Z+100 R0 FMAX

11 PATTERN DEF ROW1
(X+25 Y+33.5 D+8 NUM5 ROT+0 Z
+0)

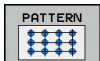


Defining a single pattern



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.** und **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.

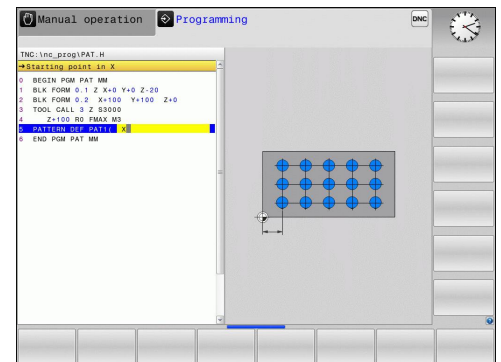


- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- ▶ **Spacing of machining positions X** (incremental): Distance between the machining positions in the X direction. You can enter a positive or negative value
- ▶ **Spacing of machining positions Y** (incremental): Distance between the machining positions in the Y direction. 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)



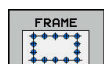
16.5 PATTERN DEF pattern definition

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.** und **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.

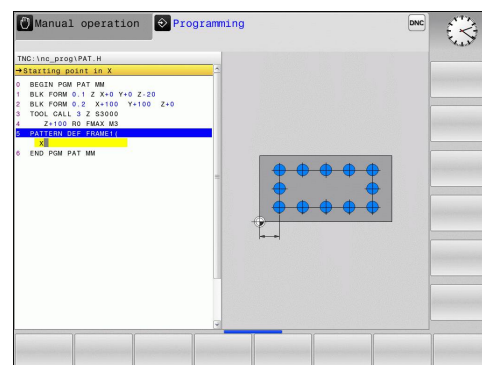


- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the frame in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the frame in the Y axis
- ▶ **Spacing of machining positions X** (incremental): Distance between the machining positions in the X direction. You can enter a positive or negative value
- ▶ **Spacing of machining positions Y** (incremental): Distance between the machining positions in the Y direction. 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

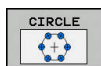
11 PATTERN DEF FRAME1
(X+25 Y+33,5 DX+8 DY+10 NUMX5
NUMY4 ROT+0 ROTX+0 ROTY+0 Z
+0)



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

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.

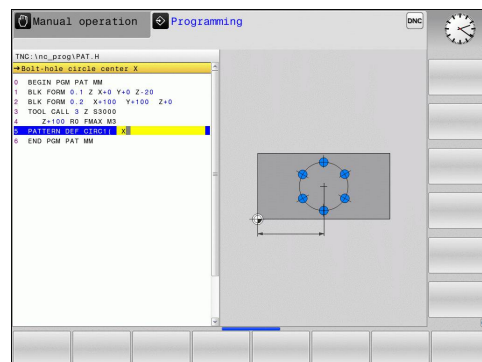


- ▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis.
- ▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis.
- ▶ **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- ▶ **Starting angle**: Polar angle of the first 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
- ▶ **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

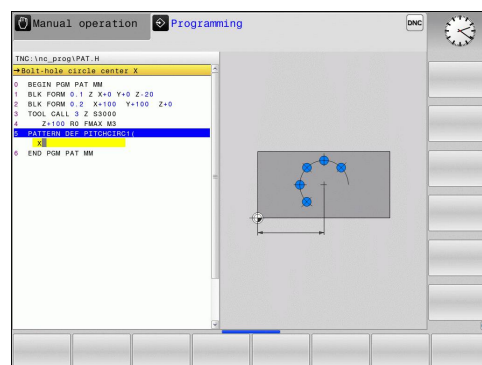
11 PATTERN DEF CIRC1
(X+25 Y+33 D80 START+45 NUM8 Z+0)



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)

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.

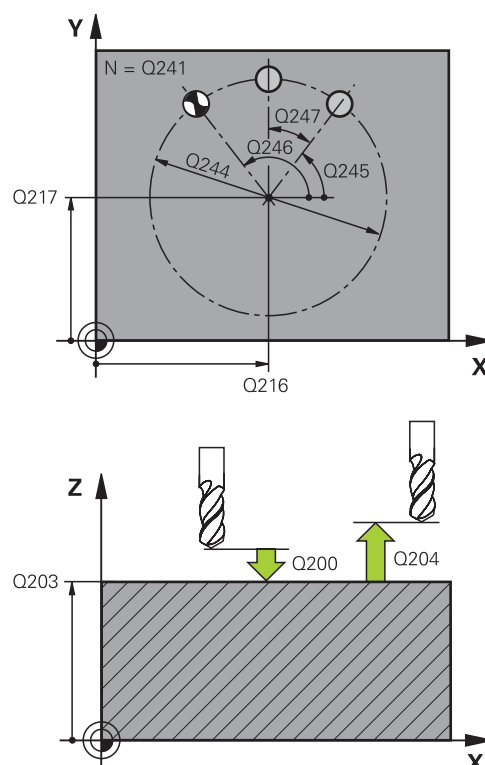
If you combine Cycle 220 with one of the fixed cycles 200 to 207 and 251, 253 and 256, the set-up clearance, workpiece surface and the 2nd set-up clearance that were defined in Cycle 220 will be effective.

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



- ▶ **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 secondary 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?**: Total number of machining positions on the 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 touch probe is to move between machining operations:
 - 0**: Move at set-up clearance between machining operations
 - 1**: Move at 2nd set-up 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)

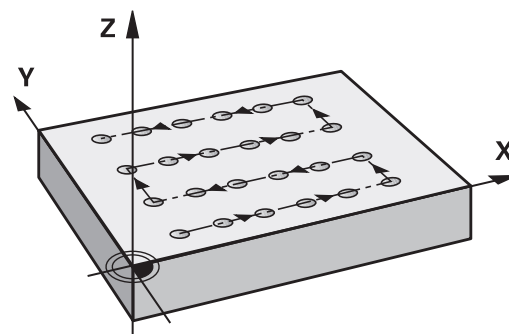
16.7 LINEAR PATTERN (Cycle 221)

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.

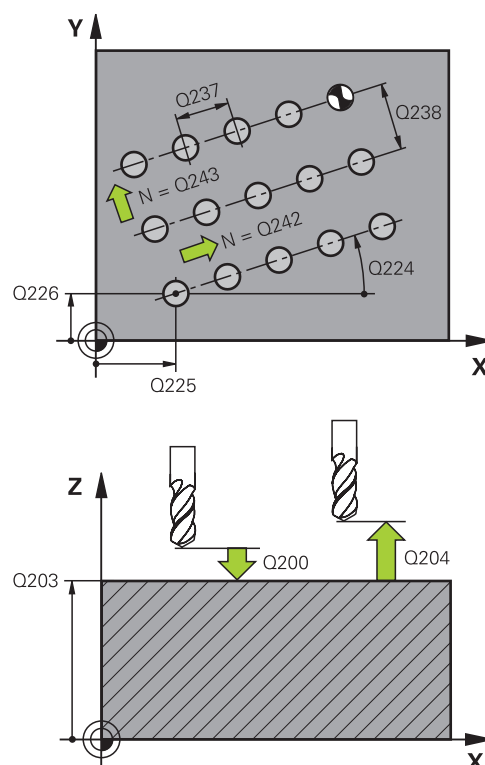
If you combine Cycle 221 with one of the fixed cycles 200 to 207 and 251, 253 and 256, the set-up clearance, workpiece surface, the 2nd set-up clearance, and the rotational position that were defined in Cycle 221 will be effective.

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 major 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 touch probe is to move between machining operations:
0: Move at set-up clearance between machining operations
1: Move at 2nd set-up 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

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



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



- ▶ Call the file manager: Press the **PGM MGT** key.

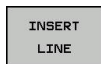
FILE NAME?



- ▶ Enter the name and file type of the point table and confirm your entry with the **ENT** key.



- ▶ 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.



- ▶ 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



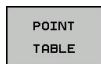
- Deactivate hiding

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:



- Press the **PGM CALL** key to call the function for selecting the 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"
```

16.8 Point tables

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**:



- ▶ 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.

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


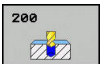

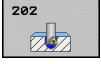
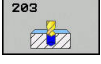
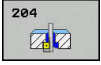




**Cycles: Drilling
cycles / thread
cycles**

17.1 Fundamentals

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	477
	200 DRILLING With automatic pre-positioning, 2nd set-up clearance	479
	201 REAMING With automatic pre-positioning, 2nd set-up clearance	481
	202 BORING With automatic pre-positioning, 2nd set-up clearance	483
	203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	486
	204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	489
	205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	493
	206 TAPPING With floating tap holder, 2nd set-up clearance, dwell time at depth	503
	207 RIGID TAPPING With thread depth and thread pitch	505
	241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition	497

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 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.



Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

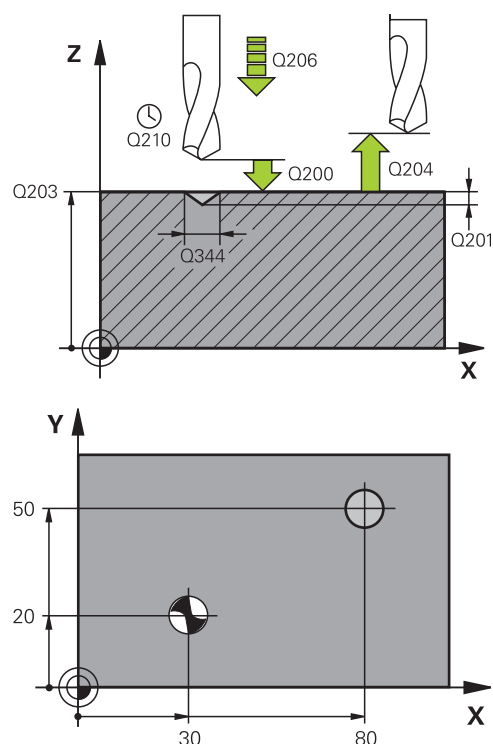
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive diameter or depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

17.2 CENTERING (Cycle 240)

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 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 during 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 total hole depth is reached.
- 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.



Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

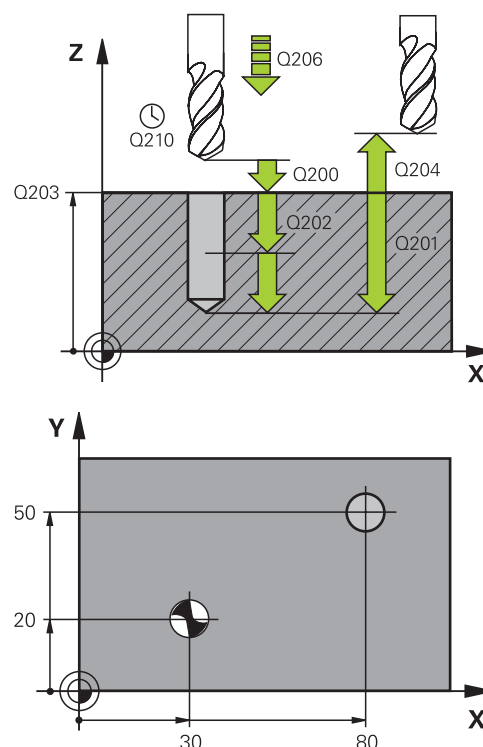
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

17.3 DRILLING (Cycle 200)

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 during 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?**: Traversing speed of the tool in mm/min during drilling. 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_r 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.



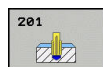
Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

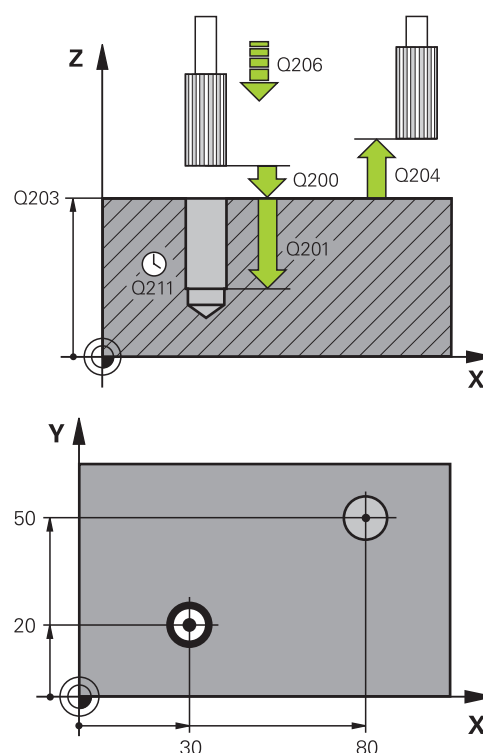
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

17.4 REAMING (Cycle 201)

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 during 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?**: Traversing speed of the tool in mm/min when retracting from the hole. 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.

17.5 BORING (Cycle 202)

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.

**Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

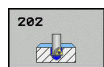
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

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

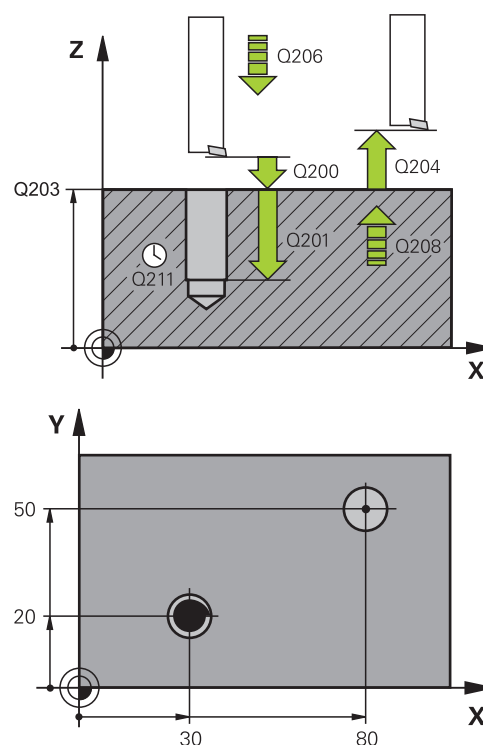
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the **Positioning with manl.data input** mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.

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

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 during 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?**: Traversing speed of the tool in mm/min when retracting from the hole. 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 retracts the tool on the hole bottom (after spindle orientation)
 - 0**: Do not retract the tool
 - 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 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	

Cycles: Drilling cycles / thread cycles

17.6 UNIVERSAL DRILLING (Cycle 203)

17.6 UNIVERSAL DRILLING (Cycle 203)

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 first plunging depth at the entered feed rate **F**.
- 3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool retracts at the retraction feed rate to the set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at **FMAX** to the set-up clearance above the first PLUNGING DEPTH.
- 4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
- 5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
- 6 The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to 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.



Danger of collision!

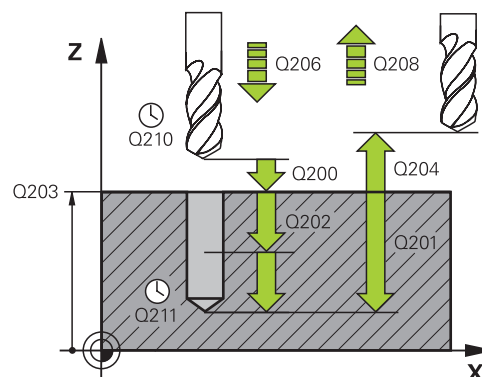
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

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

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 during 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?**: Traversing speed of the tool in mm/min during drilling. 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 removal. 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 entered **Q212 DECREMENT**, the TNC limits the plunging depth to the value for **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

17.6 UNIVERSAL DRILLING (Cycle 203)

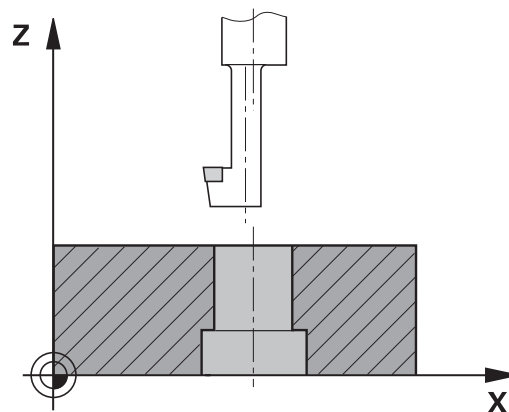
- ▶ **Q211 Dwell time at the depth?:** Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q208 Feed rate for retraction?:** Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate Q206. 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.



17.7 BACK BORING (Cycle 204)

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.

The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.

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.



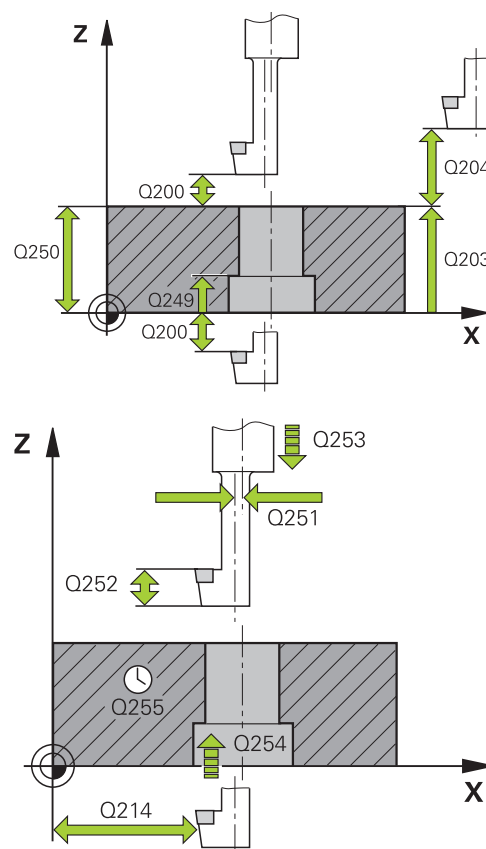
Danger of collision!

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

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 countersinking. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- ▶ **Q255 Dwell time in secs.?**: Dwell time in seconds at the top of the bore hole. 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

17.7 BACK BORING (Cycle 204)

- ▶ **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

Q253=750	;F PRE-POSITIONING
Q254=200	;F COUNTERBORING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE

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**.

17.8 UNIVERSAL PECKING (Cycle 205)

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; they are referenced to the coordinate of the workpiece surface.

**Danger of collision!**

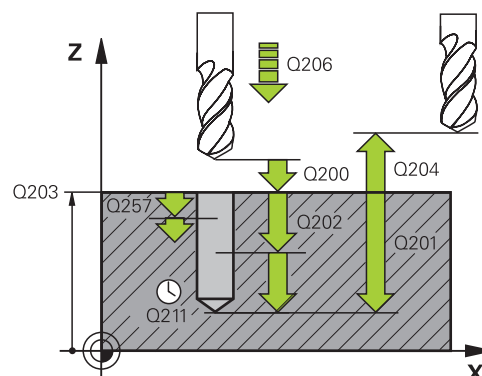
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

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

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 during 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 entered **Q212 DECREMENT**, the TNC limits the plunging depth to the value for **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

17.8 UNIVERSAL PECKING (Cycle 205)

- ▶ **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 re-approaching **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

17.9 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)

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 Then the TNC moves the tool at the defined positioning feed rate to the set-up clearance above the deepened starting point and activates the drilling speed (**M3**) and the coolant. The TNC executes the approach motion with the direction of rotation defined in the cycle, with clockwise, counterclockwise or stationary spindle.
- 3 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.
- 4 If programmed, the tool remains at the hole bottom for chip breaking.
- 5 The TNC repeats this process (3 to 4) until the programmed total hole depth is reached.
- 6 After the TNC has reached the hole depth, the TNC switches off the coolant and resets the drilling speed to the value defined for retraction.
- 7 The tool is retracted to the 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.



Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

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

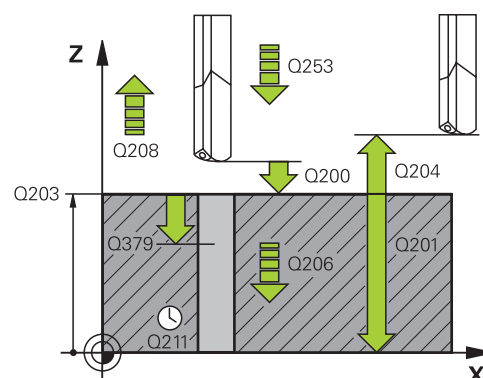
Cycles: Drilling cycles / thread cycles

17.9 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)

Cycle parameters



- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and **Q203 SURFACE COORDINATE**. Input range 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between **Q203 SURFACE COORDINATE** and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during 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 machine 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 re-approaching **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 from the hole. 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)?**: Desired direction of spindle rotation when tool moves into and retracts 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

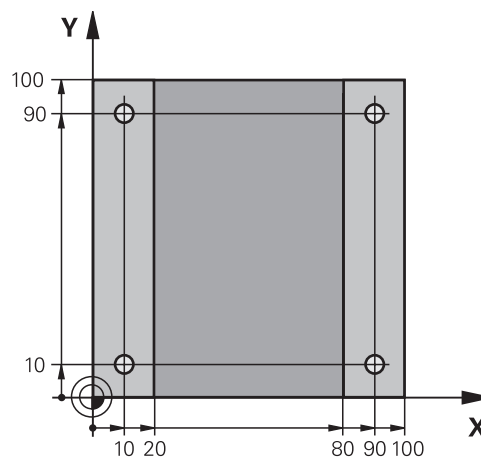
SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241) 17.9

- ▶ **Q428 Spindle speed for drilling?:** Desired speed for drilling. Input range 0 to 99999
- ▶ **Q429 M function for coolant on?:** Miscellaneous function 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 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 **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 entered **Q212 DECREMENT**, the TNC limits the plunging depth to the value for **Q205**. Input range 0 to 99999.9999

17.10 Programming Examples

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	

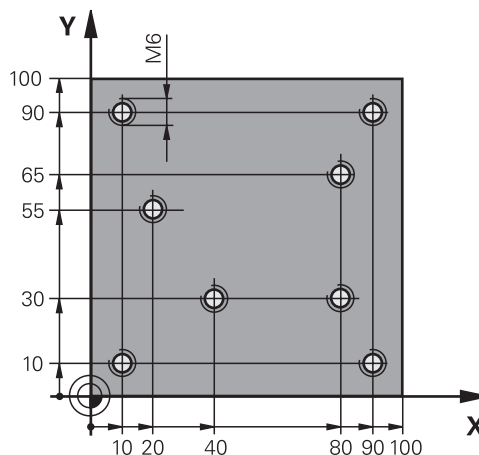
Example: Using drilling cycles in connection with PATTERN DEF

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+10 R0 F5000	Move tool to clearance height (enter a value for F): the TNC positions to the clearance height after every cycle
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=50 ;2ND SET-UP CLEARANCE	
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+10 R0 F5000	Move tool to clearance height (enter a value for F)

17 Cycles: Drilling cycles / thread cycles

17.10 Programming Examples

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=50 ;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=50 ;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.

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

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

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.



Danger of collision!

Enter in machine parameter **displayDepthErr** whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

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

17.11 TAPPING with a floating tap holder (Cycle 206)

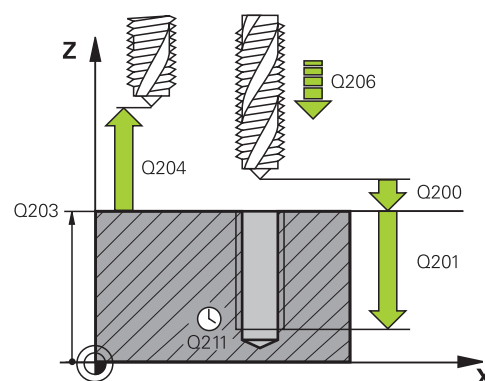
Cycle parameters



- ▶ **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 in mm/min 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: $F = S \times p$

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.

17.12 RIGID TAPPING without a floating tap holder (Cycle 207)

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.

The TNC calculates the feed rate from the spindle speed. If the feed-rate override is used during tapping, the TNC automatically adjusts the feed rate.

The feed-rate override knob is disabled.

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.

**Danger of collision!**

Enter in machine parameter **displayDepthErr** whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

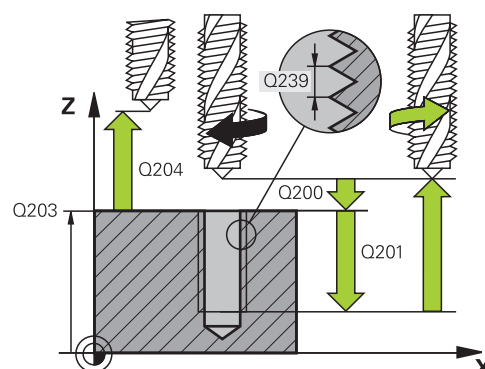
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

RIGID TAPPING without a floating tap holder (Cycle 207) 17.12

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 displays the **MANUAL TRAVERSE** soft key. After you pressed the **MANUAL TRAVERSE** soft key, you can retract the tool from 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.



When retracting the tool you can move it in the positive and negative tool axis directions. Please keep this in mind during retraction—danger of collision!

17.13 Programming Examples

17.13 Programming Examples

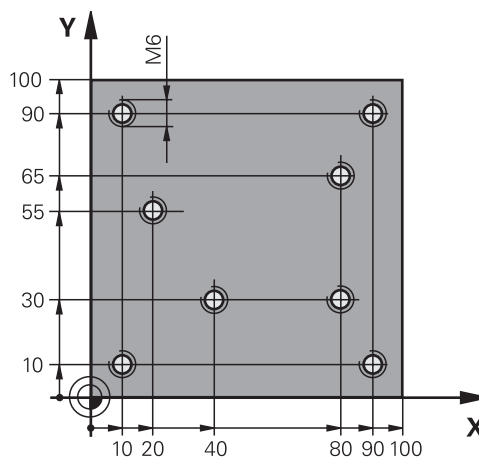
Example: Thread milling

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

Programming Examples 17.13

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

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	513
	253 SLOT MILLING Roughing/finishing cycle with selection of machining operation	517
	256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	521
	233 FACE MILLING Machining the face with up to 3 limits	525

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.

18.2 RECTANGULAR POCKET (Cycle 251)

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. Pay attention to **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.

**Danger of collision!**

Enter in machine parameter **displayDepthErr** whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

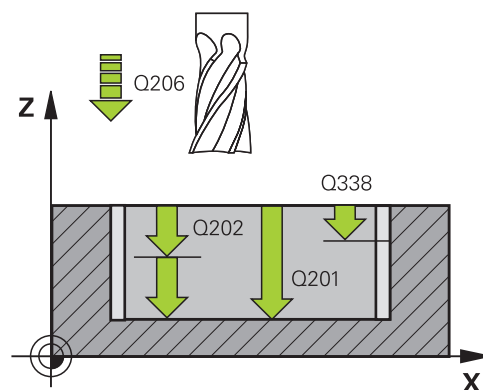
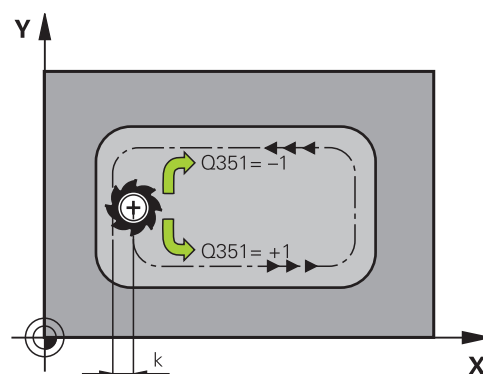
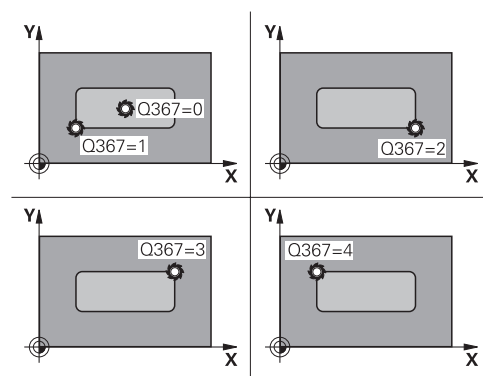
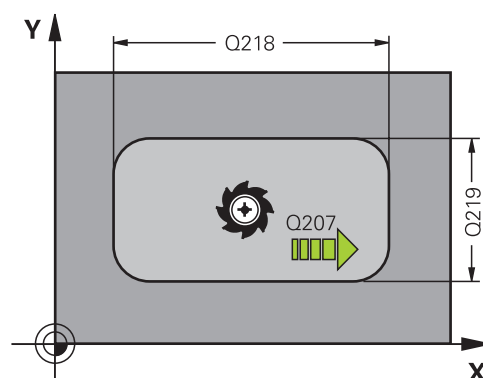
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

RECTANGULAR POCKET (Cycle 251) 18.2

Cycle parameters

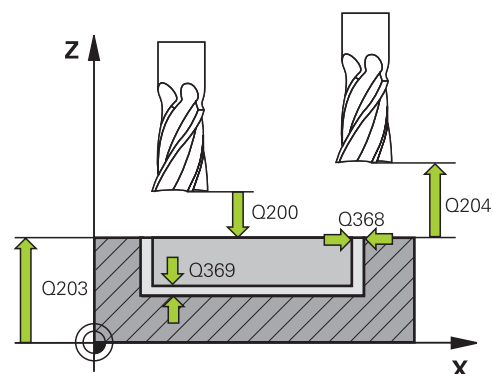


- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
 Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is deined
- ▶ **Q218 First side length?** (incremental): Pocket length, parallel 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 during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging 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;



18.2 RECTANGULAR POCKET (Cycle 251)

- ▶ **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
-1 = Up-cut (if you enter 0, climb milling is performed)
- ▶ **Q370 Path overlap factor?:** $Q370 \times \text{tool radius} = \text{stepover factor } k$. Input range: 0.1 to 1.414;



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 ;INFEEED 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.

18.3 SLOT MILLING (Cycle 253, DIN/ISO: G253)

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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

At the end of the cycle the TNC merely moves the tool in working plane back to the center of the slot; in the other working plane axis the TNC does not do any positioning. If you define a slot position not equal to 0, then the TNC only positions the tool in the tool axis to the 2nd set-up clearance. Prior to a new cycle call, move the tool back to the starting position or program always absolute traverse motions after the cycle call.

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.

**Danger of collision!**

Enter in machine parameter **displayDepthErr** whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

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

After roughing, the slot width equals the tool diameter, regardless of parameter Q219!

The diameter of the tool must be at least half the slot width.

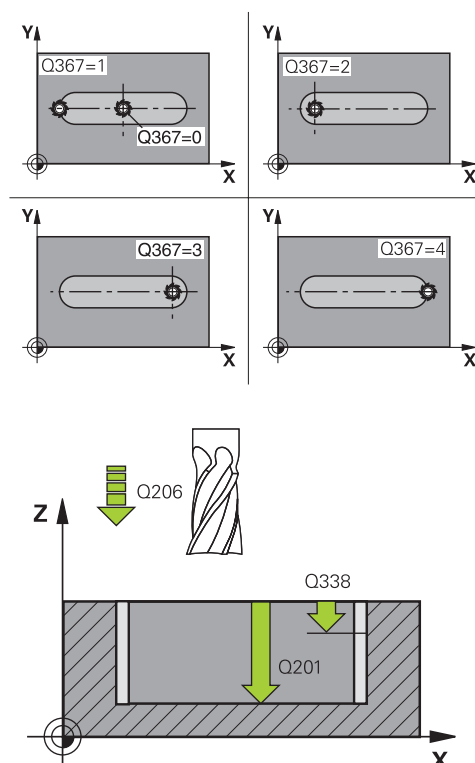
If you use a small roughing tool, then a large amount of material can remain for the finishing tool; keep this in mind when selecting your tools!

SLOT MILLING (Cycle 253, DIN/ISO: G253) 18.3

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 during milling. Input range 0 to 99999.999 alternatively **FAUTO, FU, FZ**
- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min when plunging 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 MILLING
Q206=150	;FEED RATE FOR PLNGNG
Q385=500	;FINISHING FEED RATE
Q338=5	;INFED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q351=1	;CLIMB OR UP-CUT

18.3 SLOT MILLING (Cycle 253, DIN/ISO: G253)

- ▶ **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

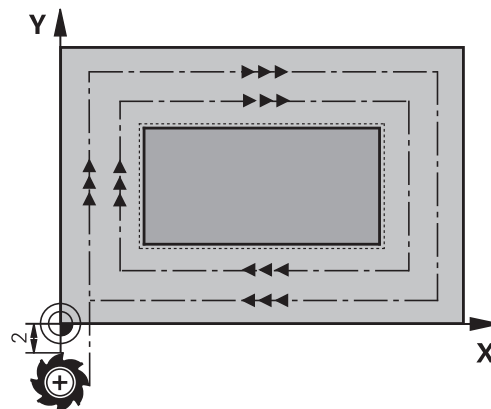
Q352=0	;PLUNGE POSITION
9 L X+50 Y+50 R0 FMAX M3 M99	

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.



18.4 RECTANGULAR STUD (Cycle 256)

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. Pay attention to **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.

**Danger of collision!**

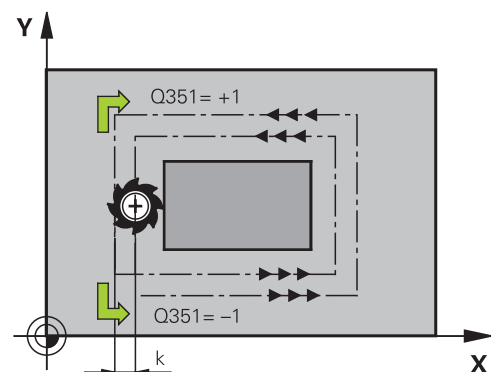
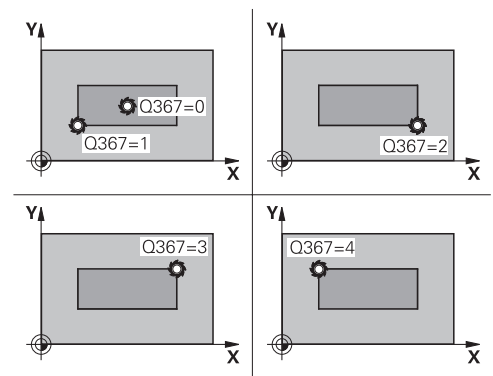
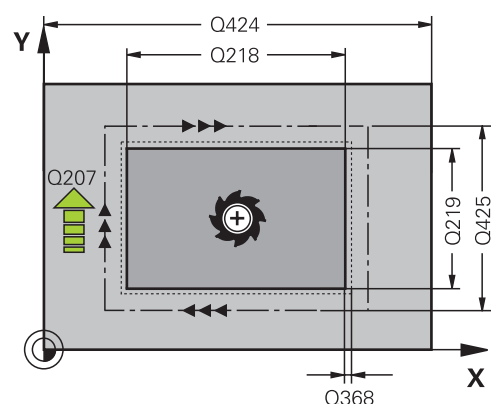
Enter in machine parameter **displayDepthErr** whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

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

Cycle parameters

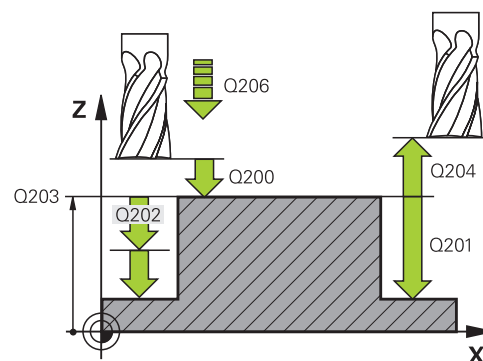


- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
 Side finishing and floor finishing are only machined when the specific allowance (Q368, Q369) is deined
- ▶ **Q218 First side length?**: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Q424 Workpiece blank side length 1?**: Length of the stud blank, 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 stud blank, 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 during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**



18.4 RECTANGULAR STUD (Cycle 256)

- ▶ **Q206 Feed rate for plunging?:** Traversing speed of the tool in mm/min when plunging to depth. Input range 0 to 99999.999; alternatively **FMAX, 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 working plane, is left over after machining. 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
-1 = Up-cut (if you enter 0, climb milling is performed)
- ▶ **Q370 Path overlap factor?:** $Q370 \times \text{tool radius} = \text{stepover factor } k$. 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:

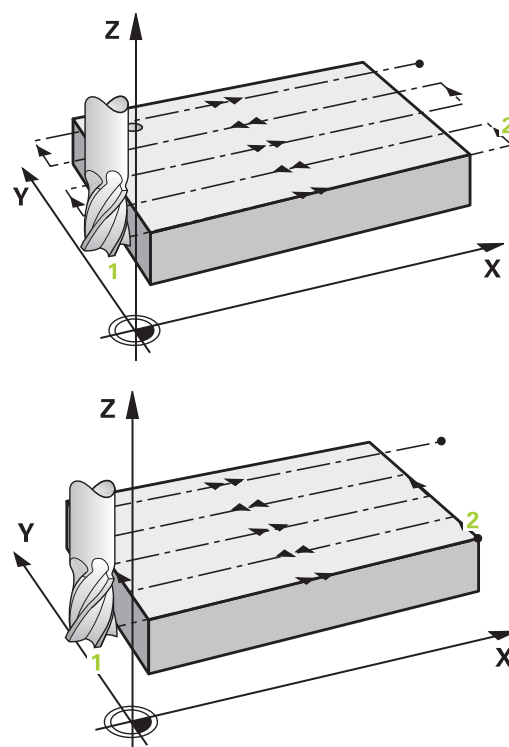
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
 - **Strategy Q389=1:** Meander machining, stepover at the edge of the surface being machined
 - **Strategy Q389=2:** The surface is machined line by line with overtravel; stepover after retracting at rapid traverse
 - **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.

18.5 FACE MILLING (Cycle 233)

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.

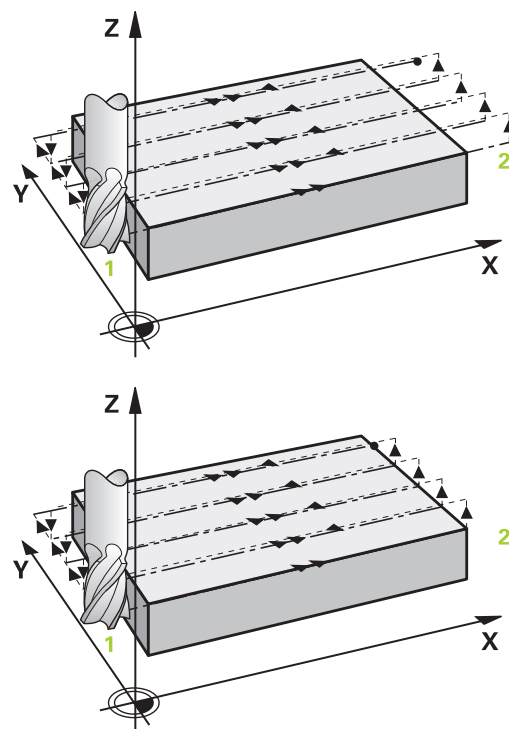
- 4 The TNC moves the tool to the end point **2** at the programmed feed rate for milling.
- 5 Then the TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and 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.

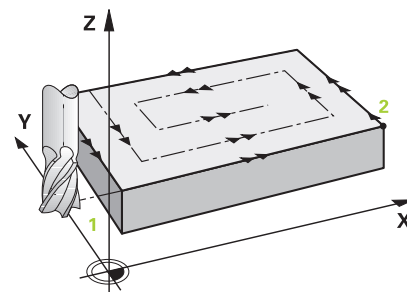
- 4 The tool subsequently advances to the end point **2** at the programmed feed rate for milling.
- 5 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at **FMAX** paraxially back to the starting point in the next line. The TNC calculates 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.



18.5 FACE MILLING (Cycle 233)

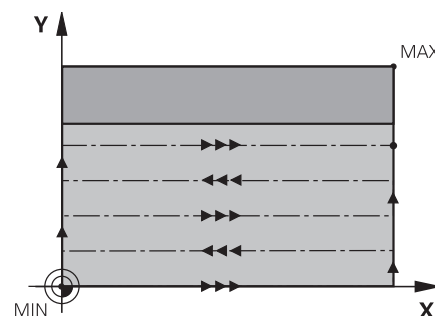
Strategy Q389=4

- 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. Pay attention to **Q204 2ND SET-UP CLEARANCE**.

Enter **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 does not run the cycle (depth = 0 has been programmed).

**Danger of collision!**

Enter in machine parameter **displayDepthErr** whether the TNC should output an error message (on) or not (off) if a positive depth is entered.

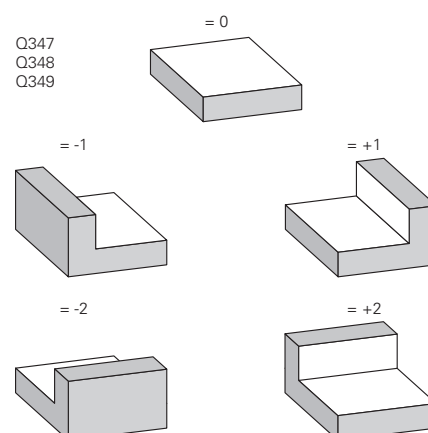
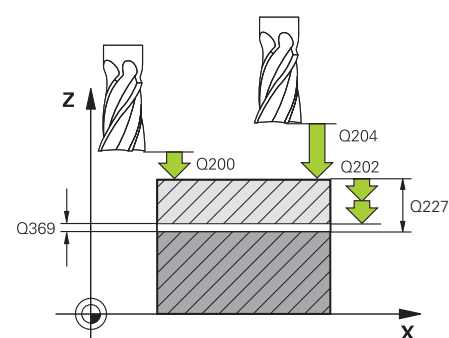
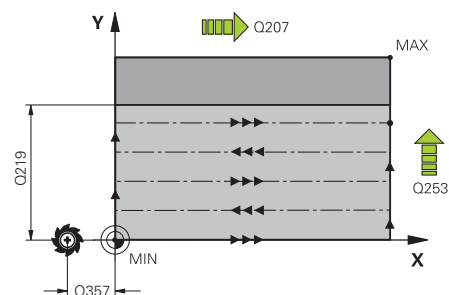
Keep in mind that the TNC reverses the calculation for pre-positioning if the starting point is less than the end point. This means that the tool moves at rapid traverse in the tool axis to set-up clearance below the workpiece surface!

18.5 FACE MILLING (Cycle 233)

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:** Machining line by line, retraction and stepover at positioning feed rate at the edge of the surface to be machined
 - 4:** Helical machining, uniform infeed 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 stepover 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



FACE MILLING (Cycle 233) 18.5

- ▶ **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 bis 1.9999.
- ▶ **Q207 Feed rate for milling?** Traversing speed of the tool in mm/min during 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): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used. Input range 0 to 99999.9999
- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range 0 to 99999.9999;

NC blocks

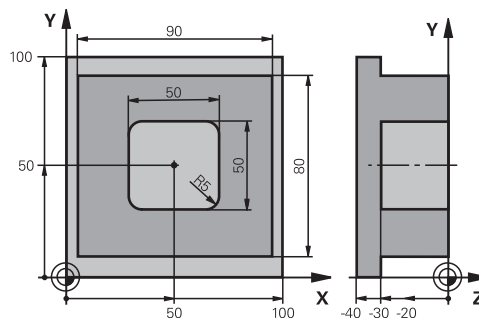
8 CYCL DEF 233 FACE MILLING	
Q215=0	;MACHINING OPERATION
Q389=2	;MILLING STRATEGY
Q350=1	;MILLING DIRECTION
Q218=120	;FIRST SIDE LENGTH
Q219=80	;2ND SIDE LENGTH
Q227=0	;STARTNG PNT 3RD AXIS
Q386=-6	;END POINT 3RD AXIS
Q369=0.2	;ALLOWANCE FOR FLOOR
Q202=3	;MAX. PLUNGING DEPTH
Q370=1	;TOOL PATH OVERLAP
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	;INFEEED FOR FINISHING
9 L X+0 Y+0 R0 FMAX M3 M99	

18.5 FACE MILLING (Cycle 233)

- ▶ **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 for 1st limit Q347
- ▶ **Q349 3rd limit?**: See parameter for 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	;INFEEED 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	

18.6 Programming Examples

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	;INFEEED 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

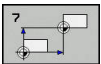

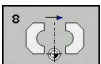
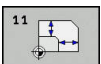
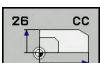
**Cycles: Coordinate
Transformations**

19.1 Fundamentals

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 For shifting contours directly within the program or from datum tables	537
	247 DATUM SETTING Datum setting during program run	543
	8 MIRRORING Mirroring contours	544
	11 SCALING FACTOR Increasing or reducing the size of contours	545
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	546

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)

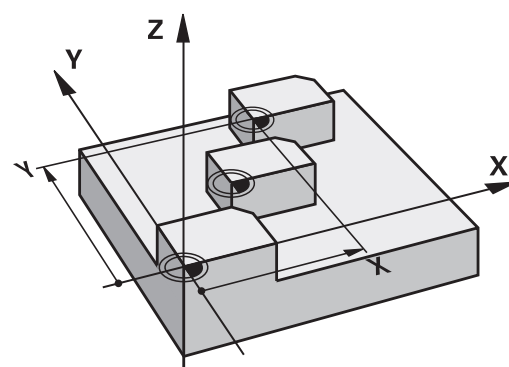
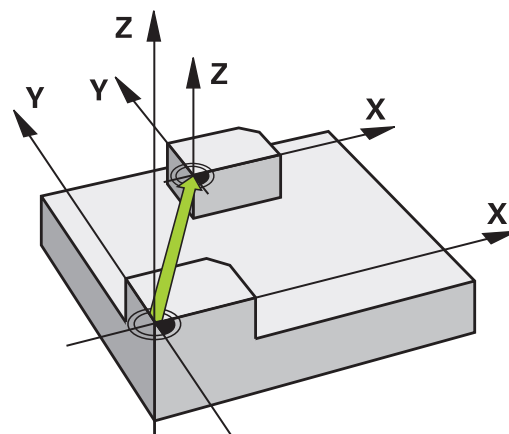
Effect

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

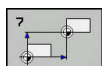
When the datum shift cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

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 manually set workpiece datum. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted. 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

Cycles: Coordinate Transformations

19.3 DATUM SHIFT with datum tables (Cycle 7)

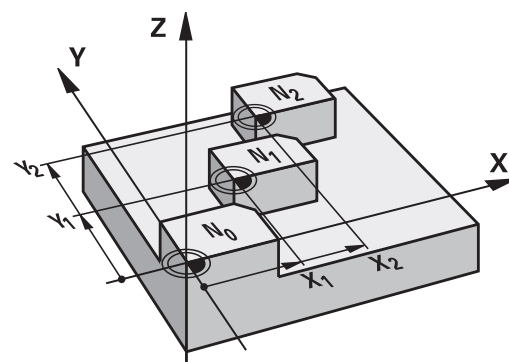
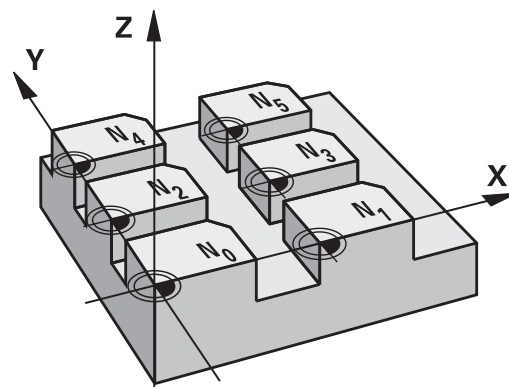
19.3 DATUM SHIFT with datum tables (Cycle 7)

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 datum points 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:**Danger of collision!**

Datums from a datum table are **always and exclusively** referenced to the current reference point (preset).



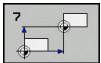
If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.

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 in the **Program run, single block** and **Program run, full sequence** operating modes to select the desired table for program run: 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

Cycles: Coordinate Transformations

19.3 DATUM SHIFT with datum tables (Cycle 7)

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.


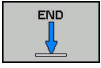


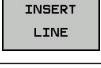
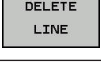
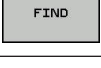
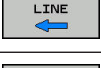
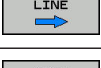
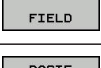
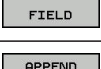
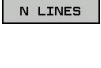
Select the datum table in the **Programming** mode of operation

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:

DATUM SHIFT with datum tables (Cycle 7) 19.3


Soft key	Function
	Select beginning of table
	Select the table end
	Go to previous page
	Go to next page
	Insert line
	Delete line
	Find
	Go to beginning of line
	Go to end of line
	Copy the current value
	Insert the copied value
	Add the entered number of lines (datums) to the end of the table

19.3

DATUM SHIFT with datum tables (Cycle 7)

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.

Manual operation

Table editing

Programming Table editing

TNC:\nc_prog\zeroshift.d

D	X	Y	Z	A	B	C
0	110.524	50.002	0	0.0	0.0	
1	200.524	50.007	0	0.0	0.0	
2	300.881	49.998	0	0.0	0.0	
3	400.994	50.001	0	0.0	0.0	
4	0.0	0.0	0.0	0.0	0.0	
5	0.0	0.0	0.0	0.0	0.0	
6	0.0	0.0	0.0	0.0	0.0	
7	0.0	0.0	0.0	0.0	0.0	
8	0.0	0.0	0.0	0.0	0.0	
9	0.0	0.0	0.0	0.0	0.0	
10	0.0	0.0	0.0	0.0	0.0	
11	0.0	0.0	0.0	0.0	0.0	
12	0.0	0.0	0.0	0.0	0.0	
13	0.0	0.0	0.0	0.0	0.0	
14	0.0	0.0	0.0	0.0	0.0	
15	0.0	0.0	0.0	0.0	0.0	
16	0.0	0.0	0.0	0.0	0.0	
17	0.0	0.0	0.0	0.0	0.0	
18	0.0	0.0	0.0	0.0	0.0	
19	0.0	0.0	0.0	0.0	0.0	

X

mm

Min. -99999.9999, max. +99999.99...

BEGIN

END

PAGE


PAGE

FIND

END

Leaving a datum table

Select a different type of file in file management and choose the desired file.



After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise the change may not be included during program run.

Status displays

In the additional status display, the TNC shows the values of the active datum shift.

19.4 DATUM SETTING (Cycle 247)

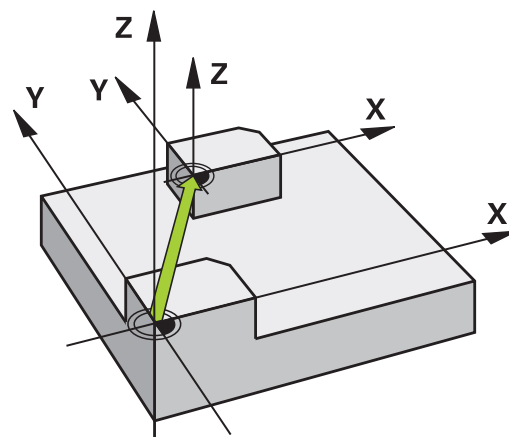
Effect

With the datum setting cycle you can activate as the new datum a preset defined in a preset table.

After a datum setting 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 datum symbol.



Please note before programming:



When activating a datum from the preset table, the TNC resets the datum shift, mirroring, scaling factor and axis-specific scaling factor.

If you activate preset number 0 (line 0), then you activate the datum that you last set in the **Manual operation** or **Electronic handwheel** operating mode. Cycle 247 is not functional in **Test run** mode.

Cycle parameters



- **Number for datum?:** Enter the number of the desired datum from the preset table. Optionally you can use the soft key **SELECT** and select the desired datum from the preset table. Input range 0 to 65535

NC blocks

13 CYCL DEF 247 DATUM SETTING

Q339=4 ;DATUM NUMBER

19.5 MIRRORING (Cycle 8)

19.5 MIRRORING (Cycle 8)

Effect

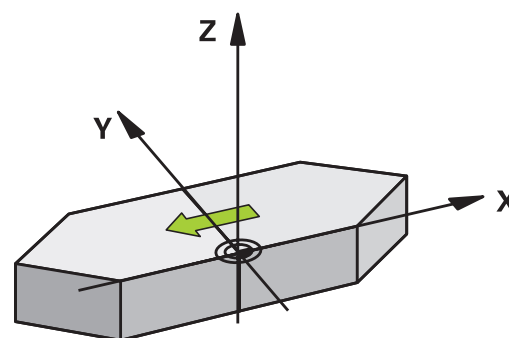
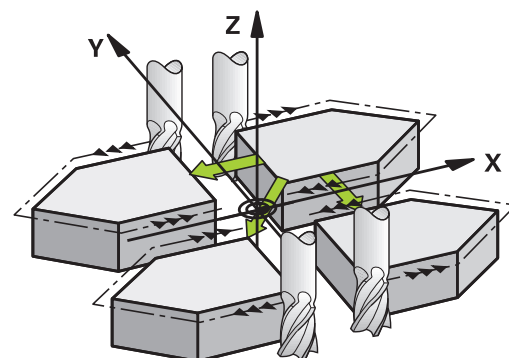
The TNC can machine the mirror image of a contour in the working plane.

The mirroring cycle becomes effective as soon as it is defined in the program. It is also effective in the **Positioning with manl.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:

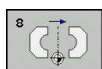
- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also “jumps” to another location.



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 the **Positioning with manl.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

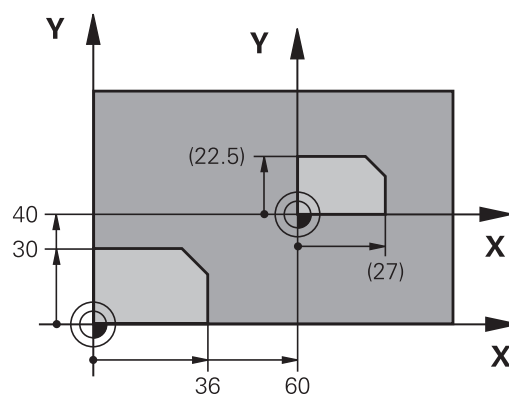
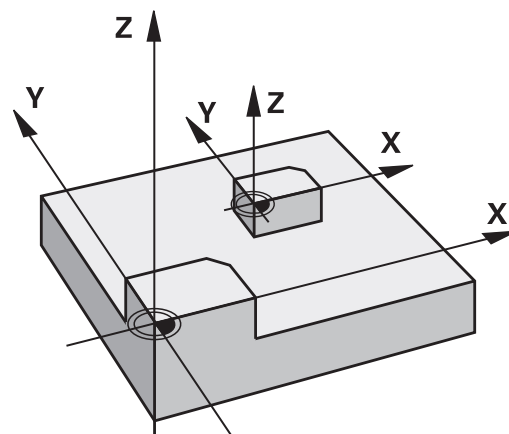
It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

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

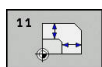
Reduction: SCL less than 1 (down to 0.000 001)

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 FACTOR
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1

Cycles: Coordinate Transformations

19.7 AXIS-SPECIFIC SCALING (Cycle 26)

19.7 AXIS-SPECIFIC SCALING (Cycle 26)

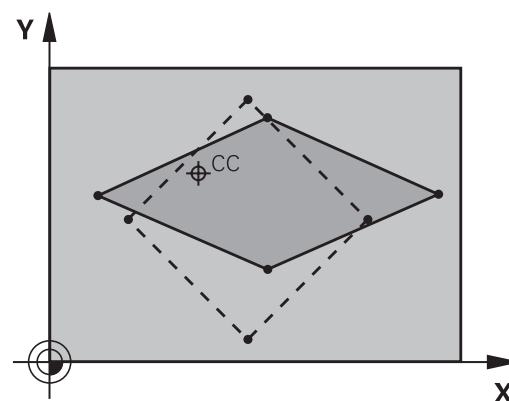
Effect

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 the **Positioning with manl.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:



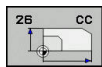
You can program each coordinate axis with its own axis-specific scaling factor.

In addition, you can enter the coordinates of a center for all scaling factors.

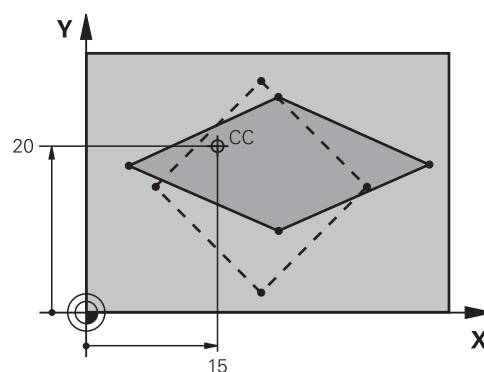
The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING FACTOR) with reference to the active datum.

AXIS-SPECIFIC SCALING (Cycle 26) 19.7

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

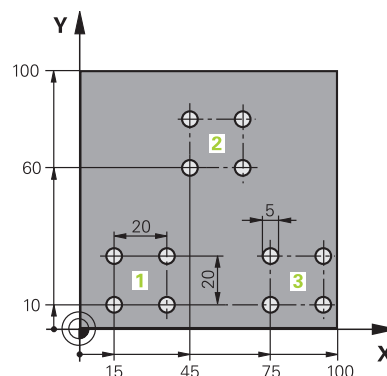
19.8 Programming Examples

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
<div><div>9</div><div></div></div>	9 DWELL TIME	553
<div><div>12</div><div>PGM CALL</div></div>	12 Program call	554
<div><div>13</div><div></div></div>	13 Oriented spindle stop	556

20.2 DWEELL TIME (Cycle 9)

Function

This causes the execution of the next block within a running program to be delayed by the programmed **DWEELL 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 DWEELL TIME

90 CYCL DEF 9.1 DWEELL 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

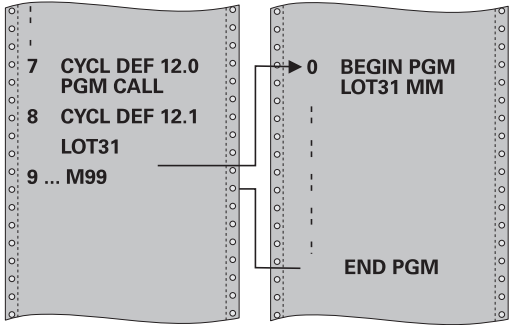
Cycles: Special Functions

20.3 PROGRAM CALL (Cycle 12)

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 with the **SELECT** soft key and select the program to be called

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

Cycles: Special Functions

20.4 SPINDLE ORIENTATION (Cycle 13)

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.

Please note while programming:

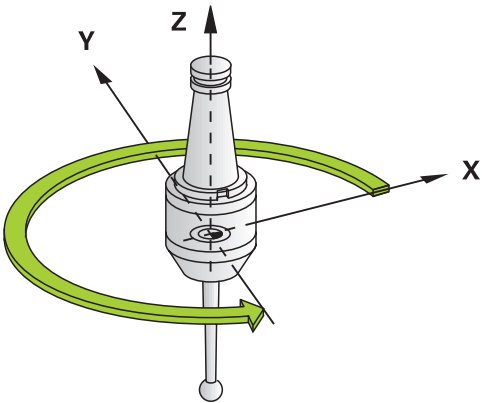


Cycle 13 is used internally for Cycles 202 and 204. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters



- **Angle of orientation:** Enter the angle referenced to the reference axis of the working plane. Input range: 0.0000° to 360.0000°



NC blocks

93 CYCL DEF 13.0 ORIENTATION

94 CYCL DEF 13.1 ANGLE 180

21

Touch probe cycles

Touch probe cycles

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

Refer to your machine manual.

The touch probe cycles are available only with option number 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 559

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).

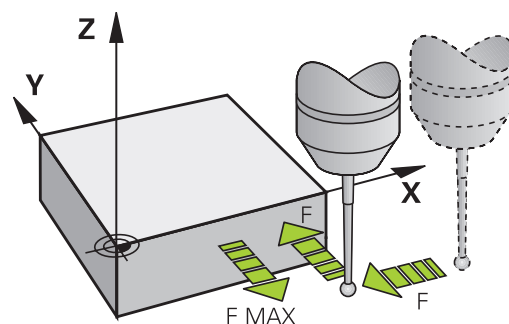
Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes

In the **Manual operation** and **Electronic handwheel** modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Setting datums

The manual touch probe cycles are described in the "Manual operation and setup".

Further Information: "Using a 3-D touch probe (option 17)", page 361

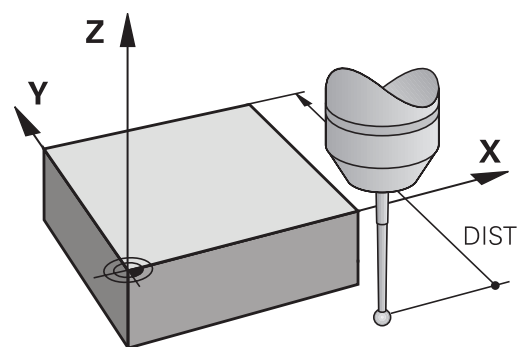


21.2 Before You Start Working with Touch Probe Cycles

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.

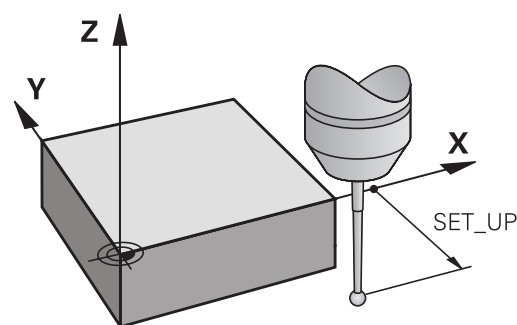
Maximum traverse to touch point: **DIST** in touch probe table

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.



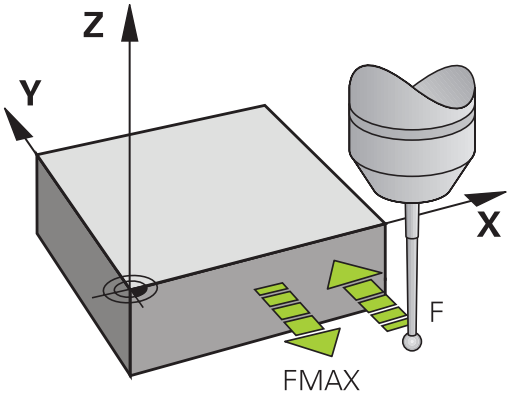
If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch probe cycles

21.2 Before You Start Working with Touch Probe Cycles

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.



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.



Danger of collision!

When running touch probe cycles, no cycles must be active for coordinate transformation (Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR** and **26 AXIS-SPECIFIC SCALING**).

Touch probe cycles with a number greater than 400 position the touch probe according to a positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (defined in the cycle), the TNC retracts the touch probe in the probe axis to the clearance height and then positions it in the working plane to the first starting position.
- 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.

Touch probe cycles

21.3 Touch probe table





21.3 Touch probe table

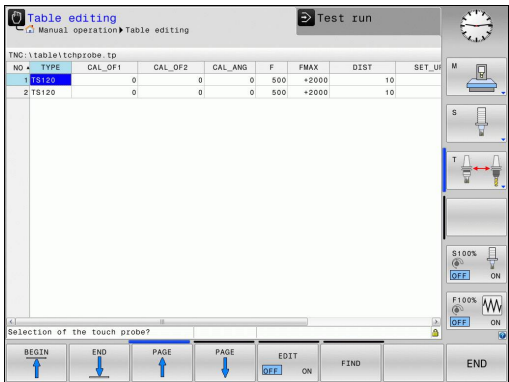
General information

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.

Editing touch probe tables

To edit the touch probe table, proceed as follows:

- 
- ▶ Operating mode: Press the **Manual operation** key
- 
- ▶ To choose the touch probe functions, press the **TOUCH PROBE** soft key. The TNC displays additional soft keys
- 
- ▶ To 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.
 - ▶ To leave 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	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 coordinated 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: <ul style="list-style-type: none"> ■ 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: <ul style="list-style-type: none"> ■ ON: Perform spindle tracking ■ OFF: Do not perform spindle tracking 	Probe oriented? Yes=ENT/No=NOENT

Touch probe cycles

21.4 Fundamentals

21.4 Fundamentals

Overview



When running touch probe cycles, Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active. HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.



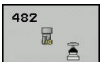



The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe. Some cycles and functions may not be provided on your machine tool. Refer to your machine manual. The touch probe cycles are available only with the Touch Probe Functions software option (option number 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	570
	Measuring the tool length, Cycle 481	573
	Measuring the tool radius, Cycle 482	575
	Measuring the tool length and radius, Cycle 483	577



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**.

Touch probe cycles

21.4 Fundamentals

Setting machine parameters



Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings > CfgTT** and **CfgTTRoundStylus**.
The TNC uses the feed rate for probing defined in the machine parameter **probingFeed** 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:

$n = \text{maxPeriphSpeedMeas} / (r \bullet 0.0063)$ 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:

$v = \text{measuring tolerance} \bullet n$ with

- v:** Feed rate for probing in mm/min
- Measuring tolerance** Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
- n:** Shaft speed [rpm]

probingFeedCalc determines the calculation of the probing feed rate:

probingFeedCalc = 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**) and the permissible tolerance (**measureTolerance1**), the sooner you will encounter this effect.

probingFeedCalc = 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 = ConstantFeed:

The feed rate for probing remains constant; the error of measurement, however, rises linearly with the increase in tool radius:

Measuring tolerance = $r \bullet \text{measureTolerance1} / 5 \text{ mm}$, where

r: Active tool radius in mm
measureTolerance1: Maximum permissible error of measurement

Touch probe cycles

21.4 Fundamentals

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 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 TNC 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 (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 L). 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)

Touch probe cycles

21.5 Calibrate the TT (Cycle 480, Option 17)

21.5 Calibrate the TT (Cycle 480, Option 17)

Cycle run

The TT is calibrated with the measuring cycle TCH PROBE 480 . The calibration process is automatic. 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 measurements.

Please note while programming:



The functioning of the calibration cycle is dependent on machine parameter **CfgToolMeasurement**. 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** > **[0]** to **[2]**.

If you change the setting of any of the machine parameters **centerPos** > **[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 datum. 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

NC blocks in new format

6 TOOL CALL 1 Z
7 TCH PROBE 480 CALIBRATE TT
Q260=+100;CLEARANCE HEIGHT

Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, Option 17) 21.6

21.6 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, 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.

Touch probe cycles

21.6 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, Option 17)

Please note while programming:



Danger of collision!

To avoid collisions, the tool must be pre-positioned before the cycle call if Q536 is set to 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.



The functioning of the calibration cycle is dependent on machine parameters. 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 the cycle. 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 cycle. 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.

Touch probe cycles

21.7 Measuring tool length (Cycle 481, Option 17)

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 datum. 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.

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 **CfgToolMeasurement**. Refer to your machine manual.

Touch probe cycles

21.8 Measuring tool radius (Cycle 482, Option 17)

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 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 datum. 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.

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 **CfgToolMeasurement**. Refer to your machine manual.

Touch probe cycles

21.9 Measuring tool length and radius (Cycle 483, Option 17)

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 datum. 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**.



To enable you to set machine-specific functions for users, your machine tool builder can define which machine parameters are available as user parameters. Furthermore, your machine tool builder can integrate additional machine parameters, which are not described in the following, into the TNC. Refer to your machine manual.

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.










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. To display the actual system names of the parameters, press the key for the screen layout key and then 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 **STORE** 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.

To exit the help text, press the **HELP** key again.

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.

22.1 Machine-specific user parameters

Parameter list

Parameter settings

DisplaySettings

Settings for screen display
Sequence of displayed axes
[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

.

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
False: Display soft key preset table

Font size with program display

- FONT_APPLICATION_SMALL**
- FONT_APPLICATION_MEDIUM**

Parameter settings

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**

22.1 Machine-specific user parameters

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

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 bis 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 [1/min]: 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

False: NC program is not stopped

22.1 Machine-specific user parameters

Parameter settings

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

22.1 Machine-specific user parameters

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 HELPOFF/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: Programmlänge, auf die die Geometrie überprüft werden soll

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

Behavior with paraxial positioning blocks

TRUE: Paraxial positioning blocks permitted

FALSE: Paraxial positioning blocks locked

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

Parameter settings

TRUE: PARAXMODE function locked

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 TNC 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 431

22.2 Connector pin layout and connection cables for data interfaces

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:

TNC		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.

Connector pin layout and connection cables for data interfaces 22.2

When using the 9-pin adapter block:

TNC		Conn. cable 355484-xx		Adapter block 363987-02		Conn. cable 366964-xx			
Male	Assignment	Female	Color	Male	Female	Male	Female	Color	Female
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	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.

22.2 Connector pin layout and connection cables for data interfaces

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	<ul style="list-style-type: none">■ Operating panel■ TFT color flat-panel display with soft keys
Program memory	<ul style="list-style-type: none">■ 2 GB
Input resolution and display step	<ul style="list-style-type: none">■ As fine as 0.1 µm for linear axes■ Up to 0.0001° for rotary axes
Input range	<ul style="list-style-type: none">■ Maximum 999 999 999 mm or 999 999 999°
Block processing time	<ul style="list-style-type: none">■ 6 ms
Axis feedback control	<ul style="list-style-type: none">■ 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	<ul style="list-style-type: none">■ Maximum 100 m (3937 inches)
Spindle speed	<ul style="list-style-type: none">■ Maximum 100,000 rpm (analog speed command signal)
Error compensation	<ul style="list-style-type: none">■ Linear and nonlinear axis error, backlash, thermal expansion■ Static friction
Data interfaces	<ul style="list-style-type: none">■ One each RS-232-C /V.24 max. 115 kilobaud■ Expanded interface with LSV-2 protocol for external operation of the TNC over the interface with HEIDENHAIN software TNCremo■ Ethernet interface 1000 BaseT■ 3 x USB (1 x front USB 2.0; 2 x rear USB 3.0)
Ambient temperature	<ul style="list-style-type: none">■ Operation: 5 °C to +45 °C■ Storage: -35 °C to +65 °C

Input formats and units of TNC functions

Positions, coordinates, chamfer lengths	-99 999.9999 to +99 999.9999 (5, 4: places before the decimal point, places 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	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ Subprograms ■ Program section repeats ■ Any desired program as subprogram
Machining cycles	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ Datum shift, mirroring ■ Scaling factor (axis-specific)
Q parameters Programming with variables	<ul style="list-style-type: none"> ■ Mathematical functions =, +, -, *, /, roots ■ Logical operations (=, ≠, <, >) ■ Calculating with parentheses ■ $\sin \alpha$, $\cos \alpha$, $\tan \alpha$, 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	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ Actual positions can be transferred directly to the NC program
Test graphics Display modes	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ 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
Datum management	■ For saving any datums
Contour, returning to	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ Calibrating the touch probe ■ Datum setting, manual ■ Tools can be measured automatically

Software options

Touch Probe Functions (option 17)

Touch probe functions	Touch probe cycles: <ul style="list-style-type: none">■ Set the datum 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	<ul style="list-style-type: none">■ HR 410: Portable handwheel■ HR 130: Panel-mounted handwheel■ HR 150: Up to three panel-mounted handwheels via handwheel adapter HRA 110
Touch probes	<ul style="list-style-type: none">■ TS 260: Triggering 3-D touch 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	MIRRORING	■	
9	DWELL TIME	■	
11	SCALING FACTOR	■	
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	DATUM SETTING	■	
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			■	298
M1	Optional program run STOP/Spindle STOP/Coolant OFF			■	418
M2	Stop program/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 1			■	298
M3	Spindle ON clockwise		■		298
M4	Spindle ON counterclockwise		■		
M5	Spindle STOP			■	
M6	Tool change/STOP program run (depending on machine parameter)/Spindle STOP			■	298
M8	Coolant ON		■		298
M9	Coolant OFF			■	
M13	Spindle ON clockwise/Coolant ON		■		298
M14	Spindle ON counterclockwise/Coolant on		■		
M30	Same function as M2			■	298
M89	Vacant miscellaneous function or cycle call, modally effective (depending on machine parameter)		■	■	456
M91	Within the positioning block: Coordinates are referenced to machine datum		■		299
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position		■		299
M94	Reduce the rotary axis display to a value below 360°		■		301
M99	Blockwise cycle call			■	456
M136	Feed rate F in millimeters per spindle revolution		■		303
M137	Reset M136				
M140	Retraction from the contour in the tool-axis direction		■		304

Index

3

3-D touch probe	
Calibrating.....	369
Using.....	361
3D Touch Probes.....	558
3-D touch probes.....	452
3-D view.....	394

A

About this manual.....	6
Accessing tables.....	264
Accessories.....	104
Actual position capture.....	118
Adding comments.....	153, 154
Additional axes.....	109
Adjusting spindle speed.....	350
Angle functions.....	238
ASCII files.....	323
Automatic tool measurement...	
184,.....	568
Axis-specific scaling.....	546

B

Back boring.....	489
Backup.....	102
Behavior after receipt of ETX....	434
Block.....	120
Delete.....	120
Block check character.....	433
Block scan	
In a point table.....	416
Bolt hole circle.....	468
Boring.....	483

C

CAD Viewer.....	209
CAD viewer and DXF converter	
screen layout.....	208
Calculating with parentheses...	275
Calculation of circles.....	239
Calculator.....	157
Centering.....	477
Circular point patterns.....	468
Code number.....	430
Condition of RTS line.....	433
Connector pin layout for data	
interfaces.....	592
Context-sensitive help.....	171
Control panel.....	82
Coordinate transformation	320, 536
Copying program sections	122, 122
Cycle.....	454
Calling.....	456
Define.....	455
Cycles and point tables.....	474

D

Data backup.....	102, 126
Data interface.....	431
Connector pin layouts.....	592
Set up.....	431
Data output on the screen.....	253
Data transfer	
File system.....	433
Software.....	435
Data transmission	
Behavior after receipt of ETX..	434
Block check character.....	433
Condition of RTS line.....	433
Data bits.....	432
Handshake.....	433
Parity.....	432
Protocol.....	432
Software TNC server.....	434
Stop bits.....	432
Datum shift.....	537
In the program.....	537
Resetting.....	322
Via the datum table.....	321
With datum tables.....	538
Datum table	
Transferring probed values.....	367
Defining local Q parameters.....	233
Defining nonvolatile Q parameters..	233
Defining the workpiece blank...	115
Dialog.....	116
Directory.....	127, 132
Copy.....	135
Create.....	132
Delete.....	136
Display handwheel.....	339
Displaying HTML files.....	142
Displaying Internet files.....	142
Display of the NC program.....	154
Display screen.....	81
DNC.....	444
Information from NC program	263
Downloading help files.....	175
Drilling.....	479, 486, 493
Drilling Cycles.....	476
Dwell time.....	317, 318, 331, 553

E

Enter spindle speed.....	193
Error message.....	166
Help with.....	166
Ethernet interface.....	437
Configuring.....	437
Connecting and disconnecting	
network drives.....	147
Connection possibility.....	437
Introduction.....	437
External access.....	423

External data transfer.....	146
-----------------------------	-----

F

FCL.....	430
FCL function.....	9
Feature Content Level.....	9
Feed rate.....	349
Adjust.....	350
Input options.....	117
Feed rate factor for plunging	
movements M103.....	302
Feed rate in millimeters per spindle	
revolution M136.....	303
File	
create.....	132
Sort.....	138
tag.....	137
File functions.....	319
File management.....	124, 127
external data transfer.....	146
File Manager	
Calling.....	129
File manager	
Copying files.....	132
Copying tables.....	134
Delete file.....	136
Directories	
Copy.....	135
Create.....	132
Directory.....	127
File type.....	124
File type	
External file types.....	126
Function overview.....	128
Overwriting files.....	133
Protect file.....	138
Rename file.....	137
Selecting files.....	130
File status.....	129
Firewall.....	443
Fluctuating spindle speed.....	315
FN14: ERROR: Displaying error	
messages.....	245, 245
FN16: F-PRINT: Output of formatted	
texts.....	249
FN18: SYSREAD: Reading system	
data.....	254
FN19: PLC: Transfer values to the	
PLC.....	262
FN20: WAIT FOR: NC and PLC	
synchronization.....	262
FN23: CIRCLE DATA: Calculate a	
circle from 3 points.....	239
FN24: CIRCLE DATA: Calculate a	
circle from 4 points.....	239
FN26: TABOPEN: Open a freely	
definable table.....	312
FN27: TABWRITE: Write to a freely	

definable table.....	313	Measuring workpieces.....	380	Principal axes.....	109
FN28: TABREAD: Read from a		Mid-program startup.....	412	Probing	
freely definable table.....	314, 314	After power failure.....	412	With end mill.....	359
FN29: PLC: Transfer values to the		Mirroring.....	544	Probing cycles.....	361
PLC.....	263	Miscellaneous functions.....	296	Manual operating mode.....	361
FN37: EXPORT.....	263	enter.....	296	Probing feed rate.....	560
FN38: SEND: Send information	263	For path behavior.....	302	Probing with a 3-D touch probe	361
Formatted output of Q parameter		For program run inspection....	298	Program.....	112
values.....	249	For spindle and coolant.....	298	Editing.....	119
Formatted output of text files...	249	Miscellaneous functions for		Opening a new program.....	115
Form view.....	311	coordinate entries.....	299	Structure.....	112
Freely definable table		Modes of Operation.....	83	Structuring.....	155
open.....	312	MOD function.....	420	Program call.....	554
write to.....	313	Exit.....	420	Any desired program as	
Fundamentals.....	108	Overview.....	421	subprogram.....	217
		Select.....	420	Via cycle.....	554
G		Move machine axes		Program defaults.....	307
Graphics.....	390	Jog positioning.....	337	Programming tool movement..	116
Display modes.....	392	Move the machine axes		Program run.....	405
With programming.....	162	with the display handwheel...	339	Execute.....	405
Magnification of details....	165	Moving the machine axes.....	336	Interrupt.....	406
Graphic settings.....	422	With axis direction keys.....	336	Mid-program startup.....	412
Graphic simulation.....	397	With the handwheel.....	338	Optional block skip.....	417
Tool display.....	397			Overview.....	405
		N		Resuming after interruption....	409
H		NC and PLC synchronization....	262	Retraction.....	410
Handwheel.....	338	NC error message.....	166	Program-section repeat.....	215
Hard disk.....	124	Nesting.....	221	Projection in three planes.....	393
Help system.....	171	Network connection.....	147	Protection zone.....	425
Help with error message.....	166	Network settings.....	437	Pulsing spindle speed.....	315
I		O		Q	
Initiated tools.....	187	Opening a BMP file.....	145	Q parameter	
Inserting and modifying blocks.	120	Opening a GIF file.....	145	Export.....	263
Interrupt machining.....	406	Opening a JPG file.....	145	programming.....	230
iTNC 530.....	80	Opening a PNG file.....	145	Transfer values to the PLC....	263
		Opening a video file.....	144	Q-Parameter	
K		Opening Excel files.....	141	Transfer values to the PLC....	262
Klartext.....	116	Opening graphic files.....	145	Q parameter programming.....	279
		Opening TXT files.....	144	Additional functions.....	244
L		Open INI file.....	144	Angle functions.....	238
Linear point patterns.....	470	Open TXT file.....	144	Calculation of circles.....	239
Load machine configuration....	449	Operating times.....	429	If-then decisions.....	240
				Mathematical functions.....	235
M		P		Programming notes.....	232
M91, M92.....	299	Part families.....	234	Q parameters.....	230, 279
Machine parameters.....	580	Path.....	127	Checking.....	242
Machine parameters for 3D touch		Pattern definition.....	462	Local parameters QL.....	230
probe.....	559	PDF Viewer.....	140	Preassigned.....	291
Machine settings.....	423	Peck drilling.....	493, 497	Residual parameters QR.....	230
Machining pattern.....	462	Plan view.....	392		
Manage datums.....	351	PLC and NC synchronization....	262	R	
Manual datum setting.....	375	Pocket table.....	190	Radius compensation.....	199
Circle center as datum.....	376	Point tables.....	472	Entering.....	200
On any axis.....	375	Positioning.....	384	Rapid traverse.....	178
Setting a center line as datum	379	With Manual Data Input.....	384	Reading out machine parameters....	288
Without a 3-D touch probe....	359	Positioning logic.....	561	288	
MDI.....	384	Preset table.....	351	Reading system data.....	254, 283
Measurement of machining		Transferring probed values....	368		
time.....	398				

Reaming.....	481	Teach In.....	118, 205	USB device	
Rectangular pocket		Test Run.....	400	Connect.....	148
Roughing+finishing.....	513	Test run		Remove.....	148
Rectangular stud.....	521	Execute.....	403	User parameters.....	580
Reference system.....	109, 109	Executing up to a certain		Using touch probe functions with	
Replacing texts.....	123	block.....	404	mechanical probes or measuring	
Resonance vibration.....	315	Test Run		dials.....	360
Restore.....	102	Overview.....	400	V	
Retraction.....	410	test run		Version number.....	430
After a power interruption.....	410	Setting speed.....	391	Version numbers.....	449
Retraction from the contour.....	304	Text file.....	323	W	
Returning to the contour.....	416	Delete functions.....	324	Window Manager.....	93
Rotary axis		Finding text sections.....	326	Wireless handwheel.....	342
Reduce display M94.....	301	Opening and exiting.....	323	Assign handwheel holder.....	446
S		Text variables.....	279	Configure.....	446
Save service files.....	170	TNCguide.....	171	Selecting transmitter power...	447
Scaling.....	545	TNCremo.....	435	Setting channel.....	447
Screen keyboard.....	152	Tool carrier management.....	327	Statistical data.....	448
Screen layout.....	81	Tool change.....	195	Working space monitoring 399, 403	
Search function.....	123	Tool compensation.....	198	Workpiece positions.....	110
Selecting the datum.....	111	Length.....	198	Write to log.....	263
Selecting the unit of measure..	115	Tool Compensation		Writing probing values	
Select kinematics.....	426	Radius.....	199	Log.....	366
Set BAUD rate.....	431	Tool data.....	180	To the datum table.....	367
Set data transmission speed....	431	Call.....	193	To the preset table.....	368
Single-lip deep-hole drilling.....	497	Delta values.....	181	Z	
Slot milling		Entering into the program.....	181	Zero point shift.....	320
Roughing+finishing.....	517	Enter into the table.....	182	Coordinate input.....	320
Software number.....	430	Tool data		ZIP archive.....	143
SPEC FCT.....	306	Initiating.....	187		
Special functions.....	306	Tool length.....	180		
Spindle orientation.....	556	Tool measurement.... 184, 564, 568			
SQL commands.....	264	Calibrate TT.....	570, 571		
Status display.....	85	Machine parameters.....	566		
Additional.....	87	Measuring tool length and			
General.....	85	radius.....	577		
Stop at.....	404	Tool length.....	573		
String parameter		Tool radius.....	575		
Converting.....	284	Tool name.....	180		
Copying a substring.....	282	Tool number.....	180		
Finding the length.....	286	Tool radius.....	180		
Testing.....	285	Tool table.....	182		
String parameters.....	279	Edit, exit.....	185		
Assign.....	280	Editing functions.....	186		
Chain-linking.....	280	Input options.....	182		
Reading system data.....	283	Tool usage file.....	426		
Structuring programs.....	155	Tool usage test.....	196, 196		
Subprogram.....	213	Touch probe cycles			
Any desired program.....	217	Manual.....	361		
Switch-off.....	335	Touch probe data.....	563		
Switch-on.....	334	Touch probe table.....	562		
T		TRANS DATUM.....	320		
Table access.....	313	Traverse limits.....	425		
Tapping		Traversing reference marks.....	334		
With a floating tap holder.....	503	Trigonometry.....	238		
Without a floating tap holder..	505	U			
		Universal drilling.....	486, 493		

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

TNC support ☎ +49 8669 31-3101

E-mail: service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103

E-mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102

E-mail: service.plc@heidenhain.de

Lathe controls ☎ +49 8669 31-3105

E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

Touch probes from HEIDENHAIN

help you reduce non-productive time and
improve the dimensional accuracy of the finished workpieces.

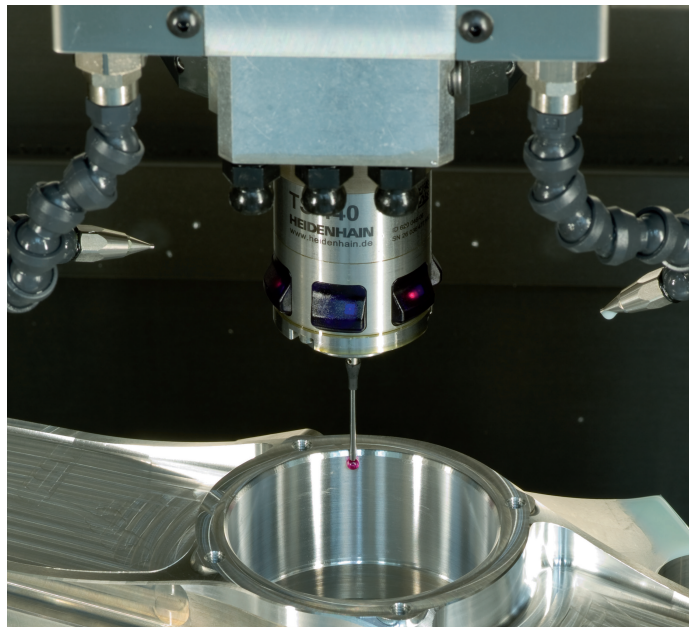
Workpiece touch probes

TS 220 Signal transmission by cable

TS 440, TS 444 Infrared transmission

TS 640, TS 740 Infrared transmission

- Workpiece alignment
- Setting datums
- Workpiece measurement



Tool touch probes

TT 140 Signal transmission by cable

TT 449 Infrared transmission

TL Contact-free laser systems

- Tool measurement
- Wear monitoring
- Tool breakage detection

