HEIDENHAIN

TNC 128

User's Manual
Conversational Programming

NC Software
771841-07

English (en)
10/2018

Controls and displays

Keys

Keys on the screen

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

Machine operating modes

Key	Function
	Manual operation
	Electronic handwheel
	Positioning with Manual Data Input
	Program Run, Single Block
	Program Run, Full Sequence

Programming modes

Key	Function
	Programming
	Test Run

Entering and editing coordinate axes and numbers

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

Tool functions

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

Managing NC programs and files, control functions

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

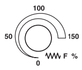
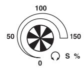
Navigation keys

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

Cycles, subprograms and program section repeats

Key	Function
CYCL DEF CYCL CALL	Define and call cycles
LBL SET LBL CALL	Enter and call subprograms and program section repeats

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed
	

Contents

1

Fundamentals

1.1 About this manual

Safety precautions

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

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

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

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

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

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

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

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

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

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

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

Have you found any errors or would you like to suggest changes?

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

tnc-userdoc@heidenhain.de

1.2 Control model, software and features

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

Control model	NC software number
TNC 128	771841-07
TNC 128 Programming Station	771845-07

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

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

- Probing functions for the 3-D touch probe

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

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

Software options

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

Additional Axis (option 0 and option 1)	
Additional axis	Additional control loops 1 and 2

Touch Probe Functions (option 17)	
Touch probe functions	**Touch probe cycles:** ■ Presetting in the **Manual operation** mode ■ Tools can be measured automatically

HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component

State Reporting Interface – SRI (option 137)	
HTTP accesses to the control status	■ Reading out the times of status changes ■ Reading out the active NC programs

Feature Content Level (upgrade functions)

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

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

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

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

Intended place of operation

The control complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

This product uses open-source software. Further information is available on the control as follows:

- Press the **MOD** key
- Select **Code-number entry**
- **LICENSE INFO** soft key

New functions 77184x-06

- New **FUNCTION COUNT** function for controlling a counter, see "Defining a counter", Page 255
- It is possible to comment out NC blocks, see "Commenting out an existing NC block", Page 122
- When multiple instances of the CAD viewer are open, they are shown somewhat smaller on the third desktop.
- With FN 16: F-PRINT, it is possible to enter references to Q parameters or QS parameters as the source and target, see "Basics", Page 201
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", Page 208
- You can also open the tool-carrier files in the file management.
- With the **ADAPT NC PGM / TABLE** function, you can also import and modify freely definable tables.
- The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs when importing a table.
- A quick search for the tool name is possible in the tool table.
- The machine tool builder can disable the setting of presets in individual axes.
- Line 0 of the preset table can also be edited manually.
- The nodes in all tree structures can be expanded and collapsed by double-clicking them.
- New icon in the status display for mirrored machining.
- Graphic settings in the **Test Run** operating mode are permanently stored.
- In the **Test Run** operating mode, you can now choose between various ranges of traverse.
- With the **TCH PROBE MONITOR OFF** soft key you can suppress touch-probe monitoring for 30 seconds.
- If the function for orienting the touch probe to the programmed probe direction is active, the number of spindle revolutions is limited when the guard door is open. In some cases, the direction of spindle rotation will change so that positioning will not always follow the shortest path.
- New machine parameter **iconPrioList** (no. 100813) for defining the order of icons in the status display.
- The machine parameter **clearPathAtBlk** (no. 124203) enables you to specify whether the tool paths will be cleared with a new BLK FORM in the **Test Run** operating mode.

Modified functions 77184x-06

- If you use locked tools, the control displays a warning in the **Programming** operating mode, see "Programming graphics", Page 135
- The **TRANS DATUM AXIS** NC syntax can also be used within a contour in the SL cycle.
- Holes and threads are shown in light blue in the programming graphics, see "Programming graphics", Page 135
- The sort order and the column widths are retained in the tool selection window when the control is switched off, see "Calling the tool data", Page 104
- If a file to be deleted does not exist, **FILE DELETE** no longer generates an error message.
- If a subprogram called with CALL PGM ends with **M2** or **M30**, the control issues a warning. The control automatically clears the warning as soon as you select another NC program, see "Programming notes", Page 168
- The time needed to paste a large amount of data into an NC program was considerably reduced.
- When you double-click a selection field of the table editor with the mouse or press the **ENT** key, a pop-up window opens.
- If you use locked tools, the control displays a warning in the **Test Run** operating mode.
- The control provides a positioning logic for returning to the contour.
- The positioning logic for returning to the contour with a replacement tool has changed.
- If the control finds a stored interruption point on restart, you can resume the machining operation from that point.
- The tool is shown in red in the graphics while it is in contact with the workpiece, and blue during air cuts.
- The positions of the sectional planes are no longer reset when a program or a new blank form is selected.
- Spindle speeds can be entered with decimal places also in the **Manual operation** mode. The control displays the decimal places when the spindle speed is < 1000.
- The control displays an error message in the header until it is cleared or replaced by a higher-priority error.
- To connect a USB stick you no longer have to press a soft key.
- The speed of setting the jog increment, spindle speed and feed rate was adjusted for electronic handwheels.
- The control automatically recognizes whether a table is to be imported or the table format is to be adapted.
- When configuration subfiles are modified, the control no longer aborts the test run, but only displays a warning.
- You can neither set nor modify a preset without having referenced the axes.
- The control issues a warning if the handwheel potentiometers are still active when the handwheel is deactivated.
- When using the HR 550 or HR 550FS handwheel, a warning is issued if the battery voltage is too low.

- The machine tool builder can define whether the **R-OFFS** offset will be taken into account for a tool with **CUT** 0.
- The machine tool builder can change the simulated tool change position.
- In the machine parameter **decimalCharakter** (no. 100805) you can define whether a period or a comma will be used as the decimal separator.

New and modified cycle functions 77184x-06

- The parameters Q215, Q385, Q369 and Q386 were added to Cycles 256 **RECTANGULAR STUD** . see "RECTANGULAR STUD (Cycle 256)", Page 374
- Changes of details in Cycle 233: Monitors the tooth length (**LCUTS**) during finishing, increases the area by Q357 in the milling direction when roughing with milling strategies 0 to 3 (provided that no limit has been set in the milling direction). see "FACE MILLING (Cycle 233)", Page 378
- The technologically outdated Cycles 1, 2, 3, 4, 5, 17, 212, 213, 214, 215, 210, 211, 230, and 231 grouped under **OLD CYCLES** can no longer be inserted using the editor. These cycles can still be executed and edited, however.
- The tool touch probe cycles, such as Cycles 480, 481 and 482, can be hidden.
- New SERIAL column in the touch probe table. see "Touch probe data", Page 419

New functions 77184x-07

- It is now possible to work with cutting data tables, see "Working with cutting data tables", Page 132
- In **Test Run** operating mode, a counter defined in the NC program is simulated, see "Defining a counter", Page 255
- An NC program you called can be edited when it has been completely executed in the calling NC program.
- With **TOOL DEF**, you can use QS parameters for entering the data, see "Entering tool data into the NC program", Page 103
- You can now use QS parameters to read from and write to freely definable tables, see "FN 27: TABWRITE – Write to a freely definable table", Page 260
- The FN16 function was expanded to include the ***** input character that can be used to write comment lines, see "Creating a text file", Page 201
- New output format for the FN16 function **%RS** that you can use to output texts without formatting, see "Creating a text file", Page 201
- The FN18 functions have been expanded, see "FN 18: SYSREAD – Reading system data", Page 208
- The new user administration enables you to create and administrate users with different access rights.
- With the new HOST COMPUTER MODE function, you can turn command over to an external host computer.
- With the **State Reporting Interface** (**SRI**), HEIDENHAIN provides a simple and reliable interface for acquiring the operating states of your machine.

- The screen layout soft keys were adapted.
- The control checks all NC programs for completeness before machining. If you attempt to start an incomplete NC program, the control aborts with an error message.
- In the **Positioning w/ Manual Data Input** operating mode, you can now skip NC blocks.
- The appearance of the **Optional program run stop** has changed.
- You can use the key between **PGM MGT** and **ERR** to toggle between screens.
- The control supports USB devices with the exFAT file system.
- If the feed rate is less than 10, the control also shows one of the decimal place that have been entered.
- In **Test Run** operating mode, the machine tool builder can define whether the tool table or the expanded tool management is opened.
- The machine tool builder defines which file types you will be able to import when using the **ADAPT NC PGM / TABLE** function.
- New machine parameter **CfgProgramCheck** (no. 129800) for defining settings for the tool usage files.

Modified functions 77184x-07

- The cutting data calculator has been improved, see "Cutting data calculator", Page 130
- The control does not run a tool change macro if neither a tool name nor a tool number is programmed in the tool call, but the same tool axis as in the previous **TOOL CALL** block, see "Calling the tool data", Page 104
- When using **SQL UPDATE** and **SQL INSERT**, the control checks the length of the table columns to be written to, see "SQL UPDATE", Page 223, see "SQL INSERT", Page 225
- When using the FN16 function, M_CLOSE and M_TRUNCATE have the same effect as far as output to the screen is concerned, see "Displaying messages on the control screen", Page 207
- In the **Test Run** operating mode, the **GOTO** key now has the same effect as in the other operating modes.
- The **ACTIVATE PRESET** soft key also updates the values of a line activated in the preset management.
- From the third desktop you can switch to any operating mode using the operating mode keys.
- The additional status display in the **Test Run** operating mode was adapted to match that of the **Manual operation** mode.
- The control allows updating of the web browser
- The screensaver glideshow was removed.
- The machine tool builder can specify which M functions to allow in the **Manual Operation** mode.
- The machine tool builder can define the default values for the L-OFFS and R-OFFS columns in the tool table.

New and modified cycle functions 77184x-07

- The REACTION column was added to the touch probe table.

2

First steps

2.1 Overview

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

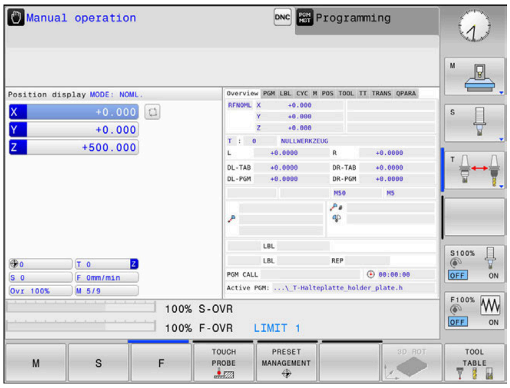
The following topics are covered in this chapter:

- Switching on the machine
- Programming the workpiece

> The following topics are covered in the User's Manual for Setup, Testing and Running NC Programs:
> - Switching on the machine
> - Graphically testing the workpiece
> - Setting up tools
> - Setting up the workpiece
> - Machining the workpiece

2.2 Switching on the machine

Acknowledging the power interruption



⚠ DANGER

Caution: Danger for the operator!

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

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

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

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

CE

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

I

- Switch on the machine control voltage
- > The control is in the **Manual operation** mode.

Depending on your machine, you may need to carry out further steps in order to run NC programs.

Further information on this topic

- Switching on the machine
  Further information: User's Manual for Setup, Testing and Running NC Programs

2.3 Programming the first part

Select operating mode

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

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

Further information on this topic

- Operating modes
  Further information: "Programming", Page 62

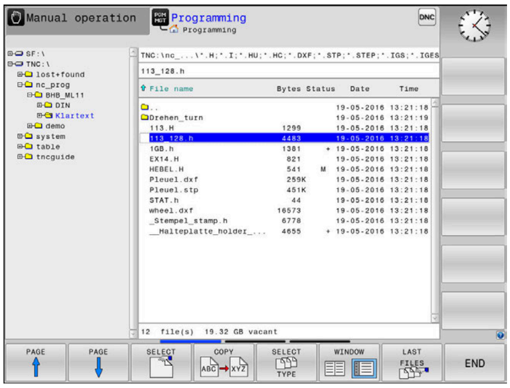
Important controls and displays

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

Further information on this topic

- Writing and editing NC programs
  Further information: "Editing an NC program", Page 75
- Overview of keys
  Further information: "Controls and displays", Page 2

Creating a new NC program / file management



PGM MGT

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

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

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

GOTO

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

ENT

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

MM

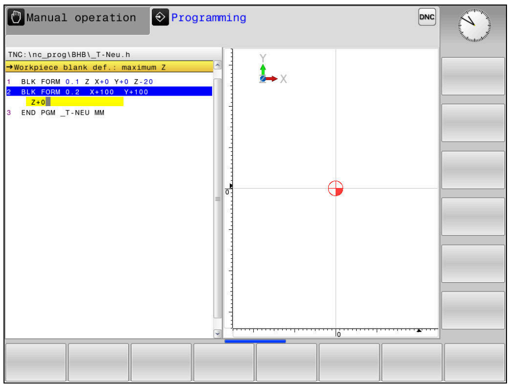
- ▶ Select the unit of measure: Press the **MM** or **INCH** soft key

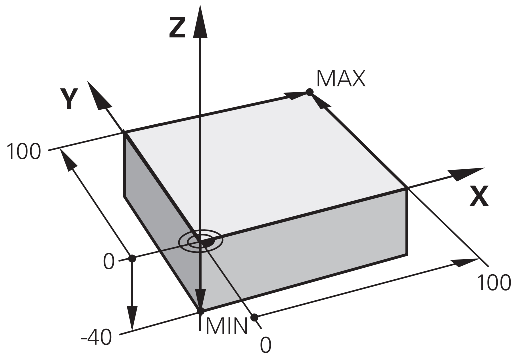
The control automatically generates the first and last NC blocks of the NC program. Afterwards you can no longer change these NC blocks.

Further information on this topic

- File management
  Further information: "File management", Page 81
- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 68

Defining a workpiece blank





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

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

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

> The control ends the dialog.

Example

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

Further information on this topic

- Define workpiece blank
  Further information: "Creating a new NC program", Page 70

Program layout

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

Recommended program layout for simple, conventional contour machining

Example

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

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

Further information on this topic

- Contour programming
  Further information: "Structure blocks in NC program", Page 112

Recommended program layout for simple cycle programs

Example

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

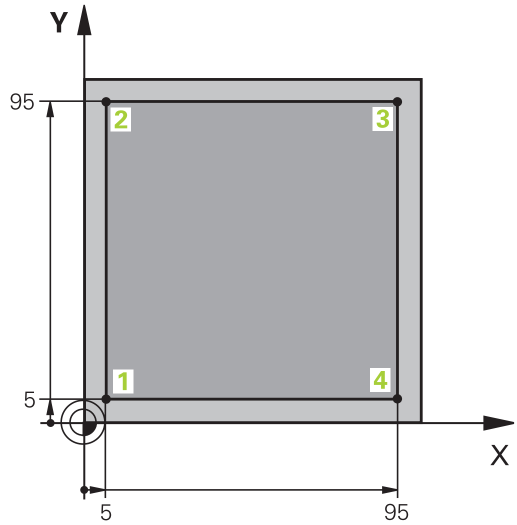
1 Call tool, define tool axis
2 Retract the tool
3 Define the machining positions
4 Define the fixed cycle
5 Call the cycle, switch on the spindle/coolant
6 Retract the tool, end the NC program

Further information on this topic

- Cycle programming
  Further information: "Fundamentals / Overviews", Page 285

Programming a simple contour

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



TOOL CALL

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

Z

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

X

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

Y

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

Z

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

X

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

Y

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

X

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

Y

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

X

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

Z

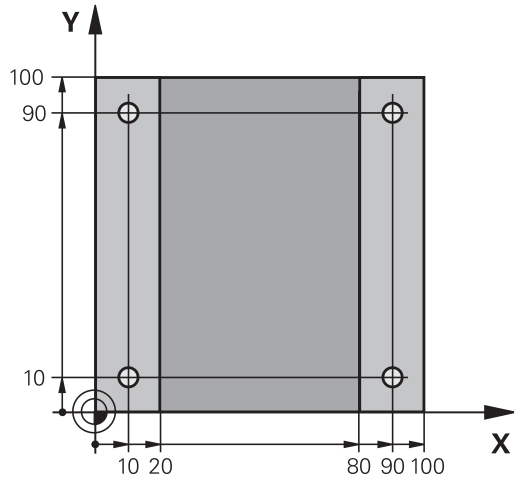
- ▶ Retract tool: Press the orange axis key Z to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Press the **ENT** key
- ▶ Confirm **Tool radius comp: R+/R-/no comp?** with the **ENT** key: Do not activate radius compensation
- ▶ **Vorschub F=?** confirm with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

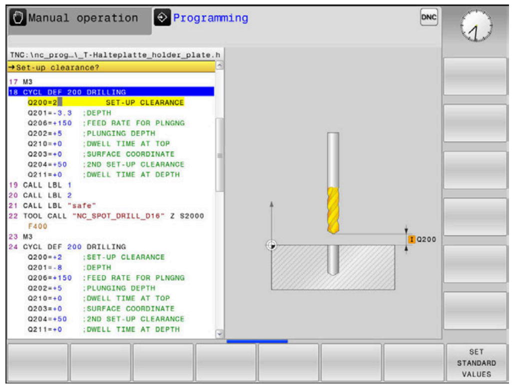
Further information on this topic

- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 68
- Programmable feed rates
  Further information: "Possible feed rate input", Page 73
- Tool radius compensation
  Further information: "Tool radius compensation with paraxial positioning blocks", Page 108
- Miscellaneous functions M
  Further information: "Miscellaneous functions for program run inspection, spindle and coolant ", Page 153

Creating a cycle program

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





TOOL CALL

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

Z

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

SPEC FCT

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

CONTOUR + POINT MACHINING

- ▶ Display the functions for point machining

PATTERN DEF

- ▶ Select the pattern definition

POINT

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

CYCL DEF

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

DRILLING/ THREAD

- ▶ Display the drilling cycles

200

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

CYCL CALL

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

CYCLE CALL PAT

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

Z

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

Example

0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S4500	Tool call
4 Z+250 R0 FMAX	Retract the tool
5 PATTERN DEF **POS1 (X+10 Y+10 Z+0)** **POS2 (X+10 Y+90 Z+0)** **POS3 (X+90 Y+90 Z+0)** **POS4 (X+90 Y+10 Z+0)**	Define the machining positions
6 CYCL DEF 200 DRILLING	Define the cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
7 CYCL CALL PAT FMAX M13	Spindle and coolant on, call the cycle
8 Z+250 R0 FMAX M2	Retract the tool, end program
9 END PGM C200 MM	

Further information on this topic

- Creating a new NC program
  Further information: "Opening and entering NC programs", Page 68
- Cycle programming
  Further information: "Fundamentals / Overviews", Page 285

3

Fundamentals

3.1 The TNC 128

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

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

HEIDENHAIN Klartext

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

You can also enter and test one NC program while another NC program is machining a workpiece.

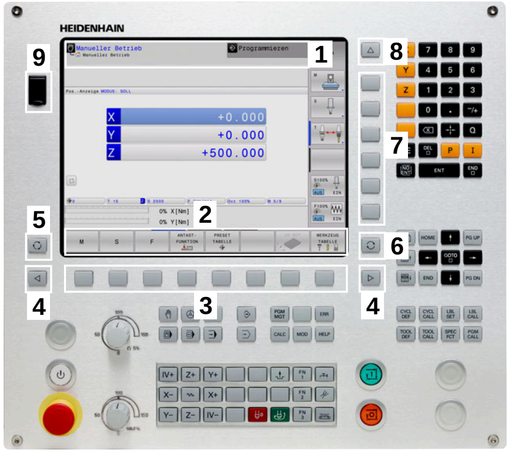
Compatibility

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

3.2 Visual display unit and operating panel

Display screen

The control is shipped with a 12.1-inch screen.



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

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

3 Soft-key selection keys

4 Keys for switching the soft keys

5 Setting the screen layout

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

7 Soft-key selection keys for machine tool builders

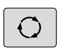
8 Keys for switching the soft keys for machine tool builders

9 USB connection

Setting the screen layout

You select the screen layout yourself. In the **Programming** operating mode, for example, you can have the control show the NC 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 the NC program blocks in one large window. The available screen windows depend on the selected operating mode.

Setting the screen layout:



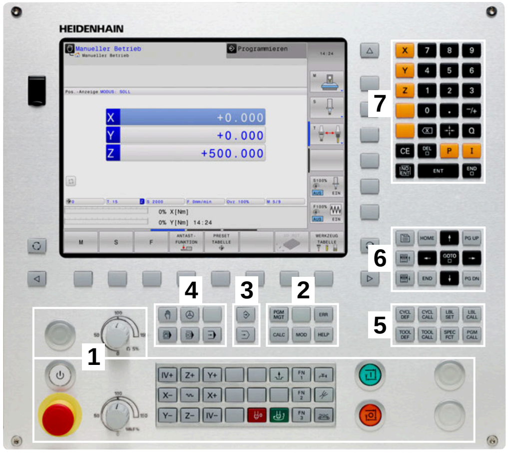
- Press the **screen layout** key: The soft-key row shows the available layout options
  Further information: "Modes of operation", Page 62

PROGRAM + GRAPHICS

- Select the desired screen layout with a soft key

Control panel

The TNC 128 is delivered with an integrated operating panel.



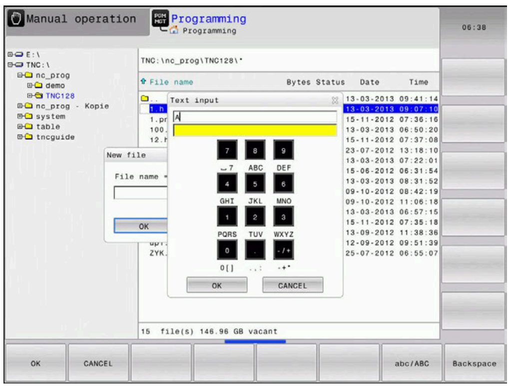
1 Machine operating panel
  More information: Machine manual
2
  - File management
  - Calculator
  - MOD function
  - HELP function
  - Show error messages
  - Toggle between the operating modes
3 Programming modes
4 Machine operating modes
5 Initiating programming dialogs
6 Navigation keys and **GOTO** jump command
7 Numerical input, axis selection and programming of positioning blocks

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

> Refer to your machine manual.
> Some machine tool builders do not use the standard HEIDENHAIN operating panel.
> External keys, e.g. **NC START** or **NC STOP**, are described in your machine manual.

Screen keypad

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



Entering text with the screen keypad

Proceed as follows to use the screen keypad:

GOTO

- Press the **GOTO** key if you want to enter letters, e.g. a program name or directory name, using the screen keypad.
- > The control opens a window in which the numeric keypad of the control is displayed with the corresponding letters assigned.

8

- Press the numerical key until the cursor is on the desired letter
- Wait until the control transfers the selected character before you enter the next character

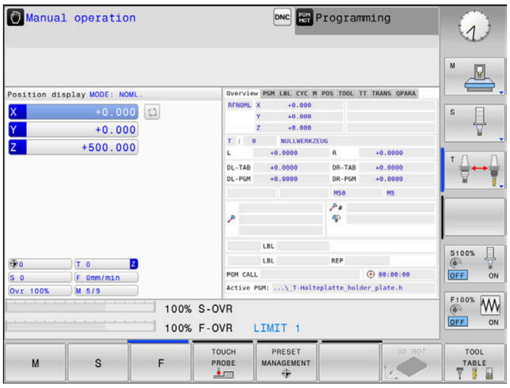
OK

- 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.

3.3 Modes of operation

Manual Operation and El. Handwheel



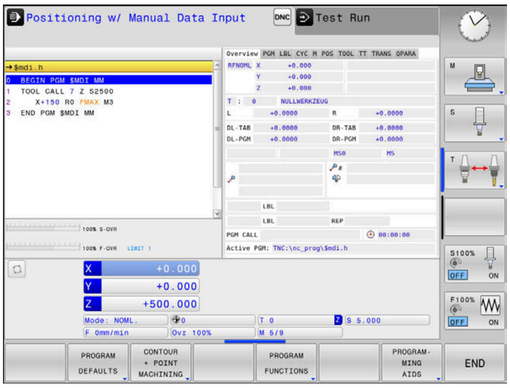
The **Manual operation** mode is required for setting up the machine tool. In this mode of operation, you can position the machine axes manually or by increments and set the presets.

The **Electronic handwheel** operating mode supports manual traverse of machine axes with the HR electronic handwheel.

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

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

Positioning with Manual Data Input

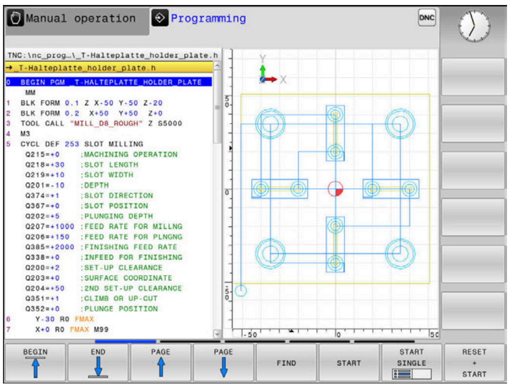


This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece

Programming



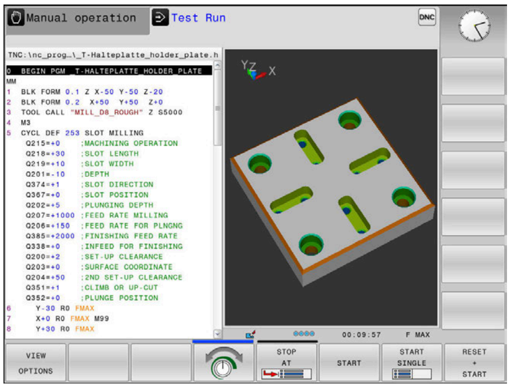
In this mode of operation you create NC programs. The various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

Soft keys for selecting the screen layout

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

Test Run

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



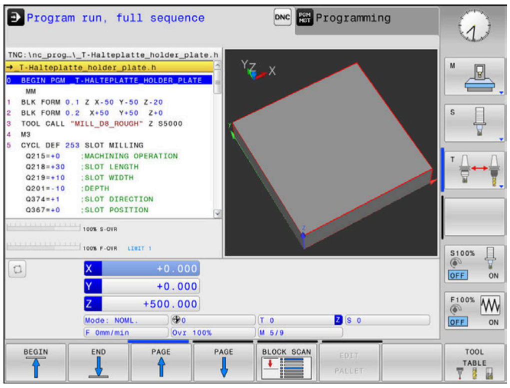
Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece

Program Run, Full Sequence and Program Run, Single Block

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

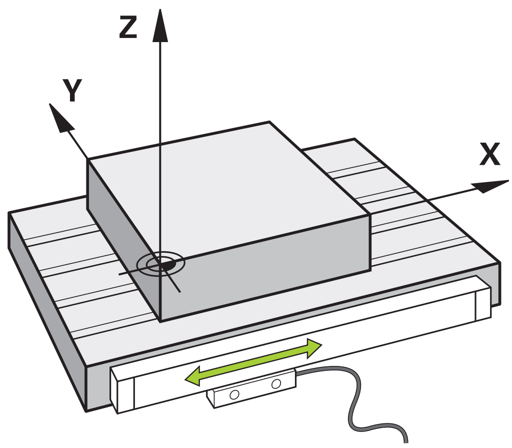


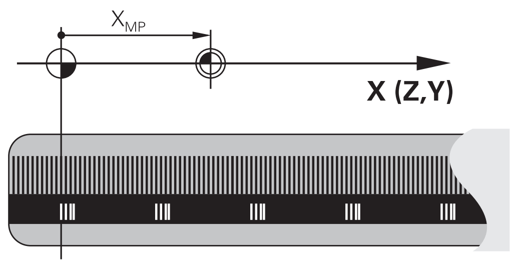
Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: structure
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece

3.4 NC fundamentals

Position encoders and reference marks





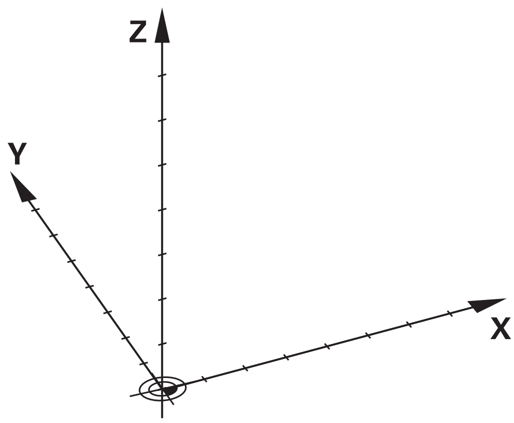
The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders.

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

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, incremental position encoders are provided with reference marks. When a reference mark is crossed over, a signal identifying a machine-based reference point is transmitted to the control. This enables the control to re-establish the assignment of the displayed position to the current machine position. For linear encoders with distance-coded reference marks, the machine axes need to move by no more than 20 mm.

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.

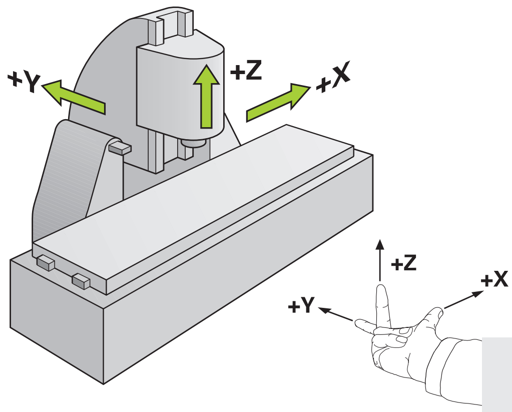
The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is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

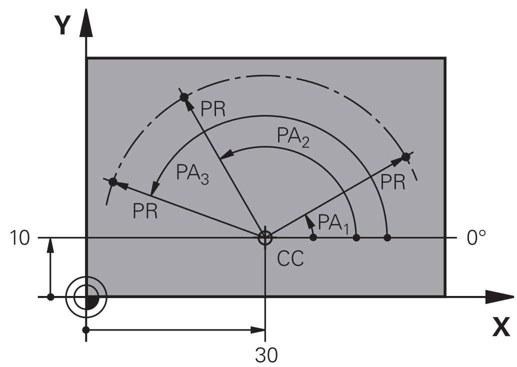
Tool axis	Principal axis	Secondary axis
X	Y	Z
Y	Z	X
Z	X	Y

Polar coordinates

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

While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at a circle center (CC), or pole. A position in a plane can be clearly defined by the:

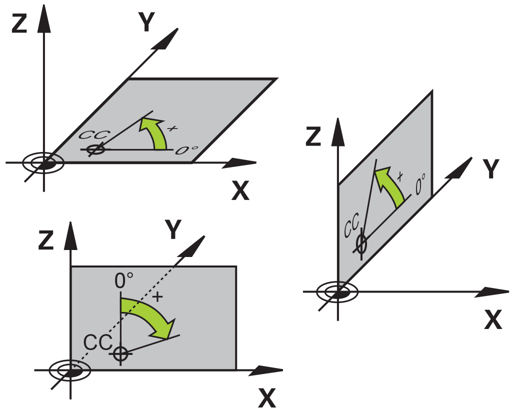
- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the value of the angle between the angle reference axis and the line that connects the circle center CC with the position.



Setting the pole and the angle reference axis

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

Coordinates of the pole (plane)	Angle reference axis
X/Y	+X
Y/Z	+Y
Z/X	+Z



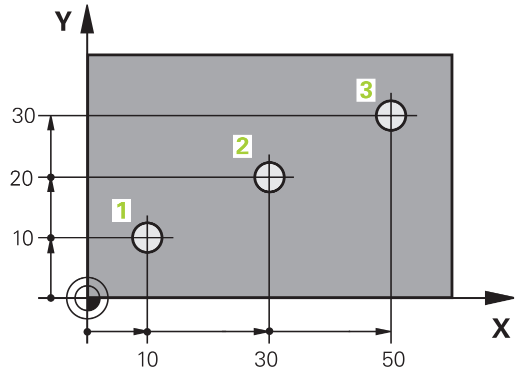
Absolute and incremental workpiece positions

Absolute workpiece positions

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

Example 1: Holes dimensioned in absolute coordinates

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



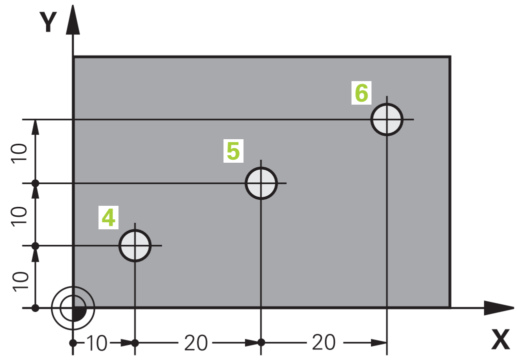
Incremental workpiece positions

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

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

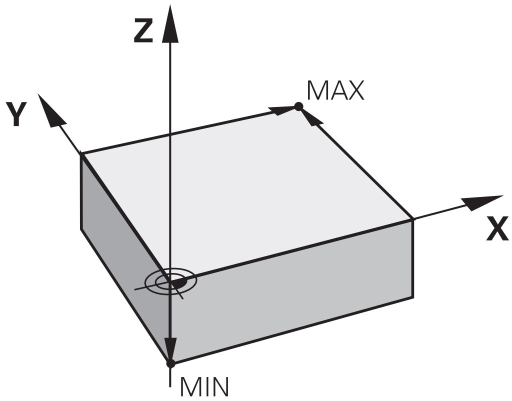
Example 2: Holes dimensioned in incremental coordinates

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



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

Selecting the datum



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

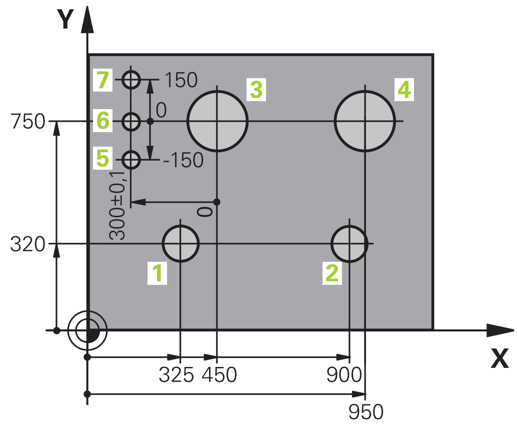
If the production drawing is dimensioned in relative presets, simply use the coordinate transformation cycles.

Further information: "DATUM SHIFT (Cycle 7)", Page 391

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

Further information: User's Manual for Setup, Testing and Running NC Programs

Example



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

3.5 Opening and entering NC programs

Structure of an NC program in HEIDENHAIN Klartext format

An NC program consists of a series of NC blocks. The illustration at right shows the elements of an NC block.

The control numbers the NC blocks of an NC program in ascending sequence.

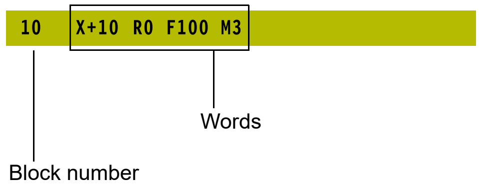
The first NC block of an NC program is identified by **BEGIN PGM**, the program name, and the active unit of measure.

The subsequent NC 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.

NC block



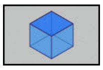
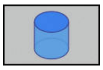
NOTICE
Danger of collision! The control does not automatically check whether collisions can occur between the tool and the workpiece. There is danger of collision during the approach movement after a tool change! ▶ If necessary, program an additional safe auxiliary position

Defining the blank: BLK FORM

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

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

The control can depict various types of blank forms:

Soft key	Function
	Define a rectangular blank
	Define a cylindrical blank

Rectangular blank

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

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

Example

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

Cylindrical blank

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

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

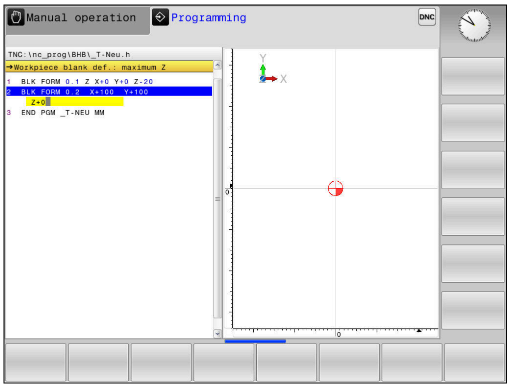
> The parameters **DIST** and **RI** or **DI** are optional and need not be programmed.

Example

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

Creating a new NC program

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



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

PGM MGT

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

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

FILE NAME = NEW.H

ENT

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

MM

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

Working plane in graphic: XY

Z

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

Workpiece blank def.: Minimum

ENT

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

Workpiece blank def.: Maximum

ENT

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

Example

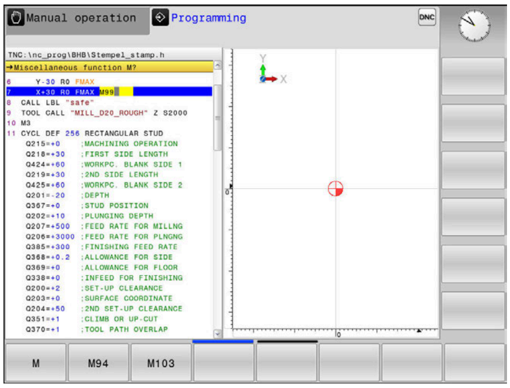
0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Spindle axis, MIN point coordinates
2 BLK FORM 0.2 X+100 Y+100 Z+0	MAX point coordinates
3 END PGM NEW MM	Program end, name, unit of measure

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

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

Programming tool movements in Klartext

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



Example of a positioning block

COORDINATES ?

X

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

ENT

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

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

ENT

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

Feed rate F=? / F MAX = ENT

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

ENT

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

MISCELLANEOUS FUNCTION M ?

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

END

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

Example

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

Possible feed rate input

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

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

Actual position capture

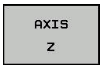
The control enables you to transfer the current tool position into the NC program,for example during

- Positioning-block programming
- Cycle programming

To transfer the correct position values, proceed as follows:

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

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



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

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

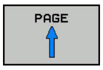
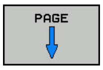
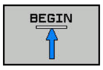
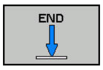
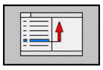
The control takes the active tool length compensation into account and always captures the coordinate of the tool tip in the tool axis.

The control keeps the soft-key row for axis selection active until the **actual position capture** key is pressed again. This behavior remains in effect even if you save the current NC block or open a new NC block with an axis key. If you have to choose an input alternative via soft key (e.g. for radius compensation), then the control closes the soft-key row for axis selection.

Editing an NC program

> You cannot edit the active NC program while it is being run.

While you are creating or editing a NC program, you can select any desired line in the NC program or individual words in an NC 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 NC block on the screen. Press this soft key to display additional NC blocks that are programmed before the current NC block No function if the NC program is fully visible on the screen
	Change the position of the current NC block on the screen. Press this soft key to display additional NC blocks that are programmed after the current NC block No function if the NC program is fully visible on the screen
 	Move from one NC block to the next NC block
 	Select individual words in an NC block
	Select a specific NC block **Further information:** "Using the GOTO key", Page 118

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

Inserting an NC block at any desired location

- Select the NC block after which you want to insert a new NC block
- Dialog initiation

Saving changes

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

- Select the soft-key row with the saving functions

STORE

- Press the **STORE** soft key
- > The control saves all changes made since the last time you saved the program.

Saving an NC program to a new file

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

- Select the soft-key row with the saving functions

SAVE AS

- Press the **SAVE AS** soft key
- > The control opens a window in which you can enter the directory and the new file name.
- Select the target directory if required with the **SWITCH** soft key
- Enter the file name
- Confirm with the **OK** soft key or the **ENT** key, or press the **CANCEL** soft key to abort

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

Undoing changes

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

- ▶ Select the soft-key row with the saving functions

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

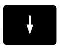
Editing and inserting words

- ▶ Select a word in an NC block
- ▶ Overwrite it with the new value
- > 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 NC blocks

- ▶ Select a word in an NC block: Press the arrow key repeatedly until the desired word is highlighted



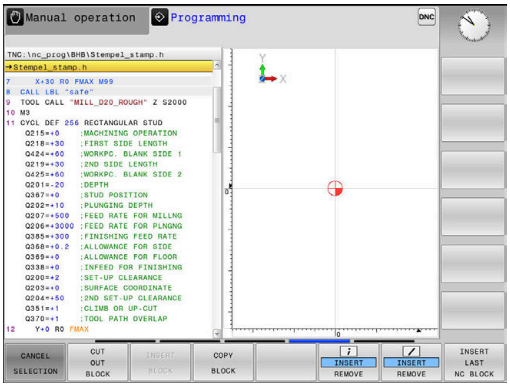
- ▶ Select an NC block with the arrow keys
  - Arrow down: search forwards
  - Arrow up: search backwards

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

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

Marking, copying, cutting and inserting program sections

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



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

To copy a program section, proceed as follows:

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

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

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

The control's search function

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

Finding any text

FIND

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

FIND

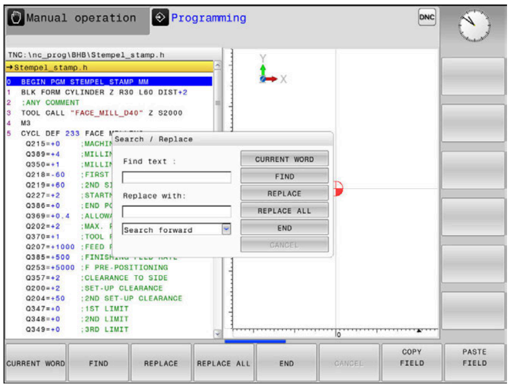
- Start the search process
- > The control moves to the next NC block containing the text you are searching for

FIND

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

END

- Terminate the search function: Press the END soft key



Finding/Replacing any text

NOTICE

Caution: Data may be lost!

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

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

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

- Select the NC block containing the word you wish to find

FIND

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

FIND

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

REPLACE

- To replace the text and then move to the next occurrence of the text, press the **REPLACE** soft key. To replace all text occurrences, press the **REPLACE ALL** soft key. To skip the text and move to its next occurrence press the **FIND** soft key

END

- Terminate the search function: Press the END soft key

3.6 File management

Files

Files in the control	Type
NC programs	
in HEIDENHAIN format	.H
Tables for	
Tools	.T
Tool changers	.TCH
Datums	.D
Points	.PNT
Presets	.PR
Touch probes	.TP
Backup files	.BAK
Dependent data (e.g. structure items)	.DEP
Freely definable tables	.TAB
Texts as	
ASCII files	.A
Text files	.TXT
HTML files, e.g. result logs of touch probe cycles	.HTML
Help files	.CHM

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

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

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

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

File names

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

File name	File type
PROG20	.H

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

The following characters are permitted:

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

The following characters have special meanings:

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

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

> The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.
>
> **Further information:** "Paths", Page 83

Displaying externally generated files on the control

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

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

Further information: User's Manual for Setup, Testing and Running NC Programs

Directories

To ensure that you can easily find your NC programs and files, we recommend that you organize your internal memory into directories (folders). You can divide a directory into further directories, which are called subdirectories. With the **-/+** key or **ENT** you can show or hide the subdirectories.

Paths

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

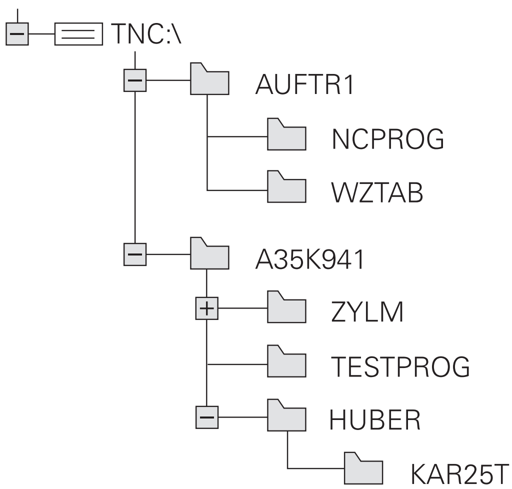
> The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

Example

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

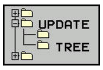
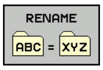
TNC:\AUFTR1\NCPROG\PROG1.H

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



Overview: Functions of the file manager

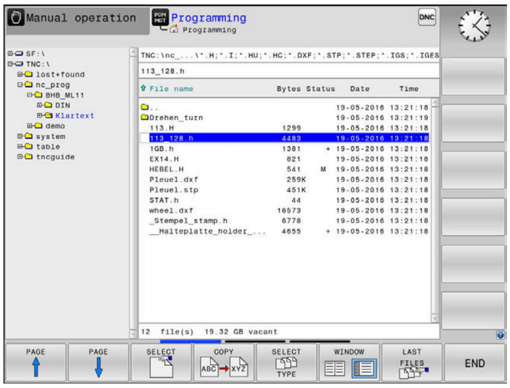
Soft key	Function	Page
COPY ABC → XYZ	Copy a single file	89
SELECT TYPE	Display a specific file type	87
NEW FILE	Create new file	89
LAST FILES	Display the last 10 files that were selected	93
DELETE	Delete a file	94
TAG	Tag a file	95
RENAME ABC = XYZ	Rename file	96
PROTECT	Protect a file against editing and erasure	97
UNPROTECT	Cancel file protection	97
ADAPT NC PGM / TABLE	Import file of an iTNC 530	See the User's Manual for Setup, Testing and Running NC Programs
	Customize table view	261
NET	Manage network drives	See the User's Manual for Setup, Testing and Running NC Programs
SELECT EDITOR	Select the editor	97
SORT	Sort files by properties	96
COPY DIR	Copy a directory	93
DELETE ALL	Delete directory with all its subdirectories	

Soft key	Function	Page
	Refresh directory	
	Rename a directory	
	Create a new directory	

Calling the file manager

PGM MGT

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



The narrow window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. A drive is the internal memory of the control. Other drives are the interfaces (RS232, Ethernet) to which you can connect a PC for example. A directory is always identified by a folder symbol to the left and the directory name to the right. Subdirectories are shown to the right of and below their parent directories. If there are subdirectories, you can show or hide them using the **-/+** key.

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

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

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

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

Selecting drives, directories and files

- 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

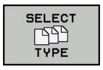
Step 1: Select drive

- Move the highlight to the desired drive in the left window
- Select a drive: Press the **SELECT** soft key, or
- Press the **ENT** key

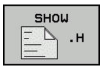
Step 2: Select a directory

- 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

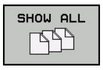
Step 3: Select a file



- Press the **SELECT TYPE** soft key



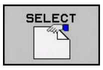
- Press the soft key for the desired file type, or



- Display all files: Press the **SHOW ALL** soft key, or

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

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



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

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

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

Creating a new directory

- Move the highlight in the left window to the directory in which you want to create a subdirectory
- Press the **NEW DIRECTORY** soft key
- Enter a directory name
- Press the **ENT** key
- Press the **OK** soft key to confirm or
- Press the **CANCEL** soft key to abort

Creating new file

- Select the directory in the left window in which you wish to create the new file
- Position the cursor in the right window
- Press the **NEW FILE** soft key
- Enter the file name with extension
- Press the **ENT** key

Copying a single file

- Move the cursor to the file you wish to copy
- Press the **COPY** soft key to select the copying function
- > The control opens a pop-up window.

Copying files into the current directory

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

Copying files into another directory

- Press the **Target Directory** soft key to select the target directory from a pop-up window
- Press the **ENT** key or the **OK** soft key
- > The control copies the file under the same name to the selected directory. The original file is retained.

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

Copying files into another directory

- Select a screen layout with two equally sized windows

In the right window

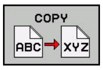
- Press the **SHOW TREE** soft key
- Move the cursor to the directory into which you wish to copy the files,

In the left window

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

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

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



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

Further information: "Tagging files", Page 95

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

Overwriting files

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

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

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

Copying a table

Importing lines to a table

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

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

NOTICE

Caution: Data may be lost!

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

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

Example

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

Proceed as follows:

- ▶ Copy this table from the external data medium to any directory
- ▶ Copy the externally created table to the existing table TOOL.T using the control's file manager.
- > The control asks you whether you want to overwrite the existing TOOL.T tool table.
- ▶ Press the **YES** soft key
- > The control will completely overwrite the current TOOL.T tool table. After this copying process the new TOOL.T table consists of 10 lines.
- ▶ Alternative: Press the **REPLACE FIELDS** soft key
- > The control overwrites 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.

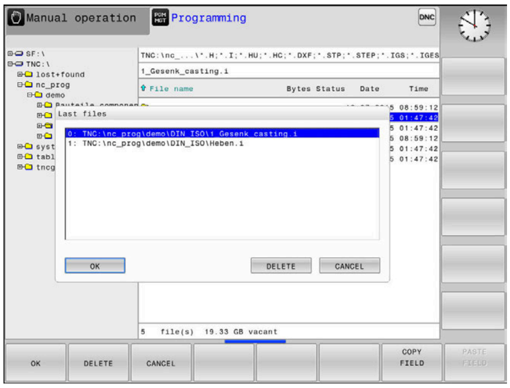
Proceed as follows:

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

Copying a directory

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

Choosing one of the last files selected



PGM MGT

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

LAST FILES

- Display the last ten files selected: Press the **LAST FILES** soft key

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

- Moves the cursor up and down within a window

OK

- Select the file: Press the **OK** soft key, or

ENT

- Press the **ENT** key

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

Deleting a file

NOTICE

Caution: Data may be lost!

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

- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the file you want to delete

- Press the **DELETE** soft key
- > The control asks whether you want to delete the file.
- Press the **OK** soft key
- > The control deletes the file.
- Alternative: Press the **CANCEL** soft key
- > The control aborts the procedure.

Deleting a directory

NOTICE

Caution: Data may be lost!

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

- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the directory you want to delete

- Press the **DELETE** soft key
- > The control inquires whether you really intend to delete the directory and all its subdirectories and files.
- Press the **OK** soft key
- > The control deletes the directory.
- Alternative: Press the **CANCEL** soft key
- > The control aborts the procedure.

Tagging files

Soft key	Tagging function
TAG FILE	Tag a single file
TAG ALL FILES	Tag all files in the directory
UNTAG FILE	Untag a single file
UNTAG ALL FILES	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:

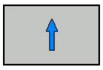
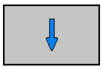
- Move the cursor to the first file

TAG

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

TAG FILE

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

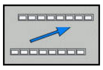



- Move the cursor to other files

TAG FILE

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

To copy tagged files:

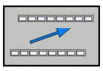


- Leave the active soft-key row

COPY ABC→XYZ

- Press the **COPY** soft key

To delete tagged files:



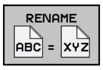
- Leave the active soft-key row

DELETE

- Press the **DELETE** soft key

Renaming a file

- Move the cursor to the file you wish to rename



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

Sorting files

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

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

Additional functions

Protecting a file and canceling file protection

- Place the cursor on the file you want to protect

- Select the additional functions:
  Press the **MORE FUNCTIONS** soft key

- Activate file protection:
  Press the **PROTECT** soft key

> The file is tagged with the "protected" symbol.

- Cancel file protection:
  Press the **UNPROTECT** soft key

Selecting the editor

- Place the cursor on the file you want to open

- Select the additional functions:
  Press the **MORE FUNCTIONS** soft key

- Select the editor:
  Press the **SELECT EDITOR** soft key
- Mark the desired editor
  - **TEXT EDITOR** for text files, e.g. **.A** or **.TXT**
  - **PROGRAM EDITOR** for NC programs **.H** and **.I**
  - **TABLE EDITOR** for tables, e.g. **.TAB** or **.T**
  - **BPM EDITOR** for pallet tables **.P**
- Press the **OK** soft key

Connecting and removing USB storage devices

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

To remove a USB device, proceed as follows:

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

Further information: User's Manual for Setup, Testing and Running NC Programs

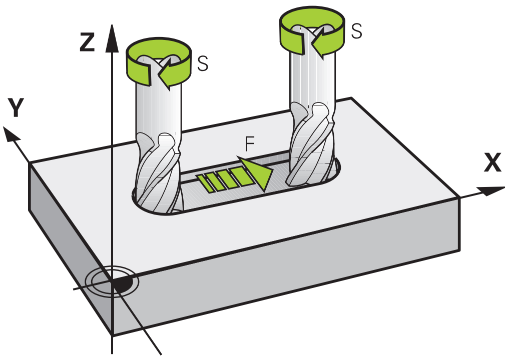
4

Tools

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

Changing during program run

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

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

Spindle speed S

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

Programmed change

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

Proceed as follows:

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 at the **Spindle speed S= ?** prompt, or switch to entry of the cutting speed by pressing the **VC** soft key

END

- Confirm your input with the **END** key

In the following cases the control changes only the speed:

- **TOOL CALL** block without tool name, tool number, and tool axis
- **TOOL CALL** block without tool name and tool number, and with the same tool axis as in the previous **TOOL CALL** block

In the following cases the control runs the tool-change macro and inserts a replacement tool if necessary:

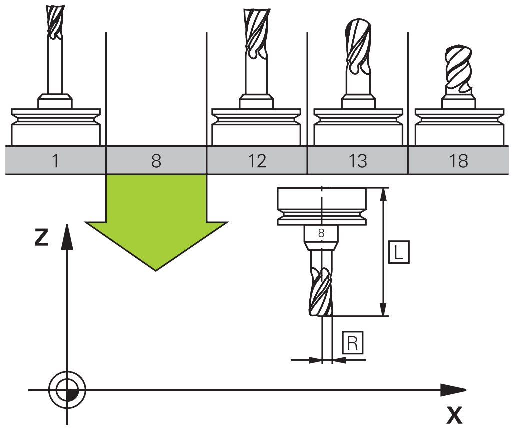
- **TOOL CALL** block with tool number
- **TOOL CALL** block with tool name
- **TOOL CALL** block without tool name or tool number, with a changed tool axis direction

Changing during program run

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

4.2 Tool data

Requirements for tool compensation



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

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

Tool number, tool name

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

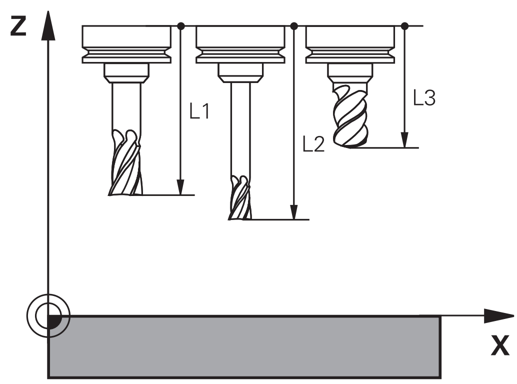
Permitted special characters: # $ % & , - _ . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z

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

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

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

Tool length L

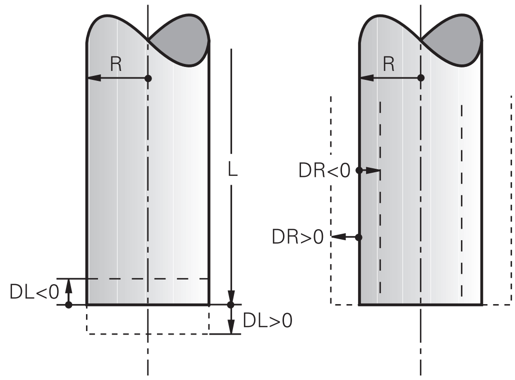


You should always enter the tool length L as an absolute value based on the tool reference point.

Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii



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

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

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

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

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

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

Entering tool data into the NC program

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

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

Proceed as follows for the definition:

TOOL DEF

- press the **TOOL DEF** key.

TOOL NUMBER

- Press the appropriate soft key
  - **Tool number**
  - **TOOL NAME**
  - **QS**
- **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
```

Calling the tool data

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

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

TOOL CALL

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

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

In the following cases the control changes only the speed:

- **TOOL CALL** block without tool name, tool number, and tool axis
- **TOOL CALL** block without tool name and tool number, and with the same tool axis as in the previous **TOOL CALL** block

In the following cases the control runs the tool-change macro and inserts a replacement tool if necessary:

- **TOOL CALL** block with tool number
- **TOOL CALL** block with tool name
- **TOOL CALL** block without tool name or tool number, with a changed tool axis direction

Tool selection in the pop-up window

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

GOTO

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

ENT

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

The following functions can be used with a connected mouse:

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

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

Tool call

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

Example

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

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

Preselection of tools

Refer to your machine manual.

The preselection of tools with **TOOL DEF** can vary depending on the individual machine tool.

If you are working with tool tables, use a **TOOL DEF** block to preselect the next tool. Simply enter the tool number, or a Q or QS parameter, or type the tool name in quotation marks.

Tool change

Automatic tool change

Refer to your machine manual.

The tool change function can vary depending on the individual machine tool.

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

Overtime for tool life

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

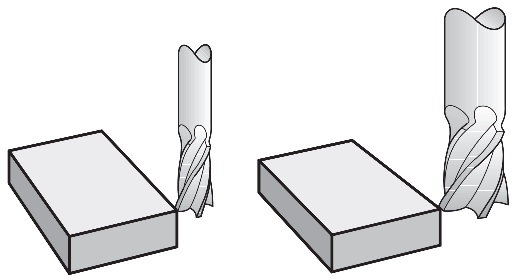
The tool condition at the end of planned tool life depends on e.g. the tool type, machining method and workpiece material. In the **OVRTIME** column of the tool table, enter the time in minutes for which the tool is permitted to be used beyond the tool life.

The machine manufacturer specifies whether this column is enabled and how it is used during tool search.

4.3 Tool compensation

Introduction

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



Tool length compensation

Length compensation becomes effective automatically as soon as a tool is called. To cancel length compensation, call a tool with the length L=0 (e.g. **TOOL CALL 0**).

NOTICE

Danger of collision!

The control uses the defined tool lengths for tool length compensation. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform a length compensation and a collision check for tools with a length of **0** and after **TOOL CALL 0**. Danger of collision during subsequent tool positioning movements!

- Always define the actual tool length of a tool (not just the difference)
- Use **TOOL CALL 0** only to empty the spindle

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

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

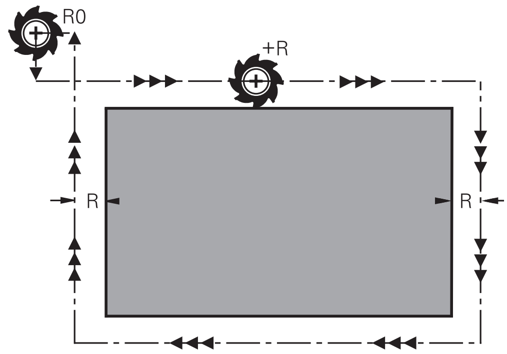
L:	Tool length **L** from **TOOL DEF** block or tool table
$\mathbf{DL}_{\text{TOOL CALL}}$:	Oversize for length **DL** in the **TOOL CALL** block
$\mathbf{DL}_{\text{TAB}}$:	Oversize for length **DL** in the tool table

Tool radius compensation with paraxial positioning blocks

The control can compensate the tool radius in the working plane with the aid of paraxial positioning blocks. You can enter the dimensions directly from the drawing without first having to convert the positions. The TNC extends or shortens the traverse path by the tool radius.

- **R+** extends the tool path by the tool radius
- **R-** shortens the tool path by the tool radius
- **R0** positions the tool using the tool center

The radius compensation is effective as soon as a tool is called and traversed with a paraxial movement in the working plane with **R+**/**R-**.



> Radius compensation is not effective for positioning movements in the spindle axis.
>
> The last selected radius compensation remains active in a positioning block that does not contain any information about radius compensation.

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

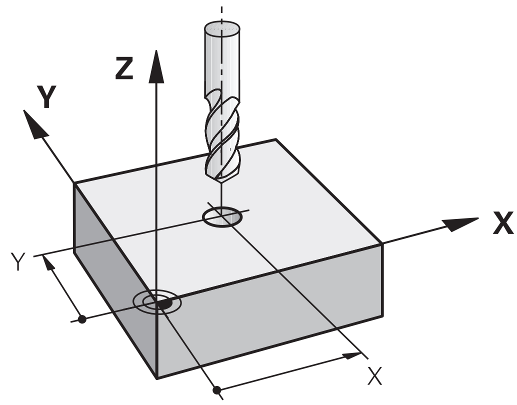
Compensation value = **R** + **DR**$_{\text{TOOL CALL}}$ + **DR**$_{\text{TAB}}$ with

R:	Tool radius **R** from **TOOL DEF** block or tool table
DR $_{\text{TOOL CALL}}$:	Oversize for radius **DR** in the **TOOL CALL** block
DR $_{\text{TAB}}$:	Oversize for radius **DR** in the tool table

Contouring without radius compensation: R0

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

Applications: Drilling and boring, pre-positioning



Entering radius compensation

Radius compensation is entered in a positioning block. Enter the coordinates of the target point and confirm your entry with the **ENT** key.

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

R+
- The TNC extends the traverse path of the tool by the tool radius

R−
- The TNC shortens the traverse path of the tool by the tool radius

ENT
- Select tool movement without radius compensation or cancel radius compensation: Press the **ENT** key

END
- Terminate the NC block: Press the **END** key

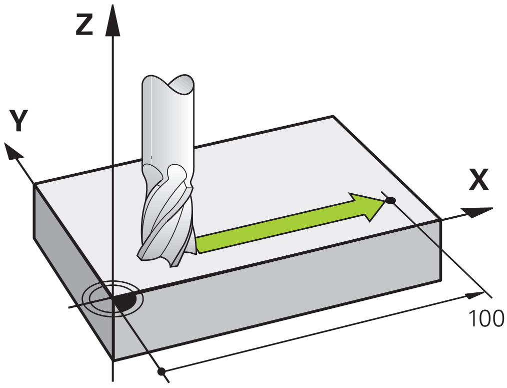
5

Programming tool - movements

5.1 Fundamentals

Structure blocks in NC program

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



X

- **Coordinates** of the end point of the movement
- **Radius compensation R+/R-/R0**
- **Feed rate F**
- **Miscellaneous function M**

Example NC block

```
6 X+45 R+ F200 M3
```

You always program the direction of tool movement. Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped.

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning can also lead to contour damage. There is danger of collision during the approach movement!

- Program a suitable pre-position
- Check the sequence and contour with the aid of the graphic simulation

Radius compensation

The control can compensate the tool radius automatically. In paraxial positioning blocks, you can select whether the control lengthens the traverse by the tool radius (R +) or shortens it (R-).

Further information: "Tool radius compensation with paraxial positioning blocks", Page 108

Miscellaneous functions M

With the control's miscellaneous functions you can affect

- the program run, e.g., a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program-section repeat. In addition, you can have an NC program call a separate NC program for execution.

Further information: "Subprograms and Program Section Repeats", Page 161

Programming with Q parameters

Instead of programming numerical values in an NC program, you enter markers called Q parameters. 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.

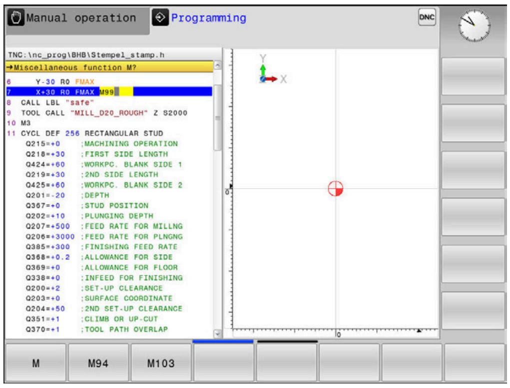
Further information: "Programming Q Parameters", Page 181

5.2 Tool movements

Programming tool movements for workpiece machining

Create an NC block with the axis keys

Use the orange axis keys to initiate the dialog. The control asks you successively for all the necessary information and inserts the program block into the NC program.



Example—programming a straight line

X

- Select the axis key you want to use for the positioning movement, e.g. **X**

COORDINATES?

- **10** Enter the coordinate of the end point, e.g. 10

ENT

- Press the **ENT** key

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

R0

- Select radius compensation, e.g. by pressing the **R0** soft key
- > The tool moves without compensation.

Feed rate F=? / F MAX = ENT

- **100** Enter the feed rate, e.g. 100 mm/min. (For programming in inches: Entry of 100 corresponds to a feed rate of 10 inches/min.)

ENT

- Press the **ENT** key

F MAX

- As an alternative, move at rapid traverse: press the **FMAX** soft key

F AUTO

- As an alternative, traverse with the feed rate defined in the **TOOL CALL** block: Press the **F AUTO** soft key

MISCELLANEOUS FUNCTION M?

- **3** (the miscellaneous function **M3** switches on the spindle)

ENT

- The control ends this dialog with the **ENT** key

The program-block window displays the following line:

6 X+10 R0 FMAX M3

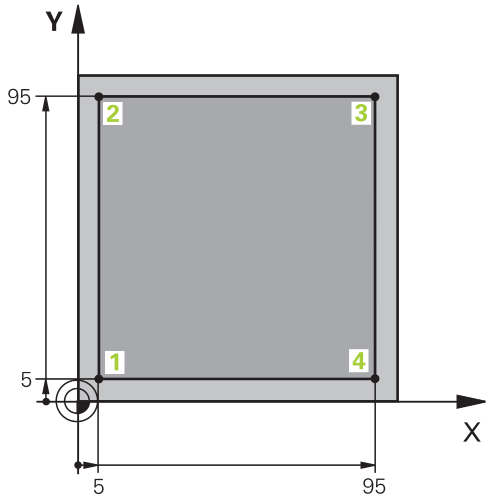
Capture actual position

You can also generate a positioning block by using the **ACTUAL-POSITION-CAPTURE** key:

- In the **Manual operation** mode, move the tool to the position to be captured
- Select the **Programming** operating mode
- Select the NC block after which you want to insert the NC block

- Press the **Actual-Position-Capture** key
- > The control generates an NC block.
- Select the desired axis, e.g. by pressing the **ACT. POS. X** soft key
- > The control loads the actual position and ends the dialog.

Example: Linear movement



0 BEGIN PGM LINEAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Call the tool in the spindle axis and with the spindle speed S
4 Z+250 R0 FMAX	Retract the tool in the spindle axis at rapid traverse FMAX
5 X-10 R0 FMAX	Pre-position the tool
6 Y-10 R0 FMAX	Pre-position the tool
7 Z+2 R0 FMAX	Pre-position the tool
8 Z-5 R0 F1000 M13	Move to working depth at feed rate F = 1000 mm/min
9 X+5 R- F500	Contour approach
10 Y+95 R+	Move to point 2
11 X+95 R+	Move to point 3
12 Y+5 R+	Move to point 4
13 X-10 R0	Close the contour and retract
14 Z+250 R0 FMAX M30	Retract the tool, end program
16 END PGM LINEAR MM	

6

Programming Aids

6.1 GOTO function

Using the GOTO key

Jumping with the GOTO key

Use the **GOTO** key to jump to a specific location in the NC program, regardless of the active operating mode.

Proceed as follows:

GOTO □

- ▶ Press the **GOTO** key
- \> The control opens a pop-up window.
- ▶ Enter a number

N LINES ↓

- ▶ Select the jump statement by soft key, e.g. move down the number of lines entered

The control provides the following options:

Soft key	Function
N LINES ↑	Move up the number of lines entered
N LINES ↓	Move down the number of lines entered
GOTO LINE NUMBER	Jump to the block number entered

> Use the **GOTO** function only during programming and testing of NC programs. Use the block scan function during program run.
>
> **Further information:** User's Manual for Setup, Testing and Running NC Programs

Quick selection with the GOTO key

With the **GOTO** key, you can open the Smart Select window that makes it easy for you to select special functions or cycles.

Proceed as follows to select special functions:

SPEC FCT

- ▶ Press the **SPEC FCT** key

GOTO □

- ▶ Press the **GOTO** key
- \> The control displays a pop-up window showing a structural view of the special functions
- ▶ Select the desired function

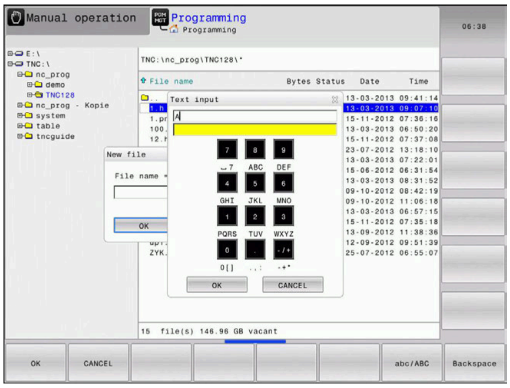
Further information: "Defining a cycle using the GOTO function", Page 289

Opening the selection window with the GOTO key

When the control provides a selection menu, you can use the **GOTO** key to open the selection window. This allows you to view the available entries.

6.2 Screen keypad

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



Entering text with the screen keypad

Proceed as follows to use the screen keypad:

GOTO

- ▶ Press the **GOTO** key if you want to enter letters, e.g. a program name or directory name, using the screen keypad.
- > The control opens a window in which the numeric keypad of the control is displayed with the corresponding letters assigned.

8

- ▶ Press the numerical key until the cursor is on the desired letter
- ▶ Wait until the control transfers the selected character before you enter the next character

OK

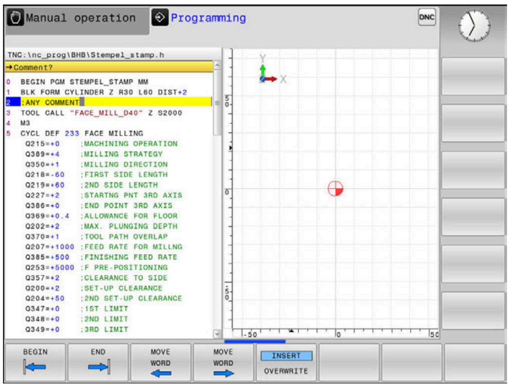
- ▶ Use the **OK** soft key to load the text into the open dialog field

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.

6.3 Display of NC programs

Syntax highlighting

The control displays syntax elements with various colors according to their meaning. Color-highlighting makes the NC programs easier to read and clearer.



Color highlighting of syntax elements

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

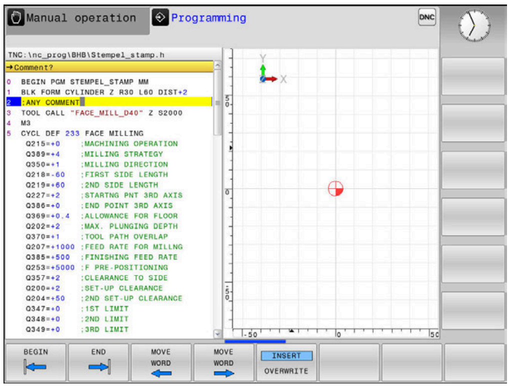
Scrollbar

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

6.4 Adding comments

Application

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



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

You can add comments in different ways.

Add comments

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

SPEC FCT
- Press the **SPEC FCT** key

PROGRAM-MING AIDS
- Press the **PROGRAMMING AIDS** soft key

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

Entering comments during programming

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

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

Inserting comments after program entry

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

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

Entering a comment in a separate NC block

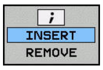
> To use this function you will need a alphabetic keyboard connected via USB.

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

Commenting out an existing NC block

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

- Select the NC block to be commented out

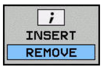


- Press the **INSERT COMMENT** soft key
- > The control inserts a semicolon **;** at the beginning of the block.
- Press the **END** key

Changing a comment for an NC block

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

- Select the comment block you want to change

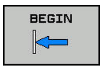
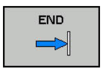
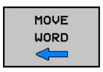
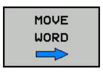
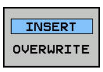


- Press the **REMOVE COMMENT** soft key

Alternative:

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

Functions for editing of the comment

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

6.5 Freely editing an NC program

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

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

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

Free syntax input using the control's integrated text editor

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

PGM MGT

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

MORE FUNCTIONS

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

SELECT EDITOR

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

OK

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

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

Free syntax input using the ? key in the NC editor

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

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

⇧ ? /

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

END

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

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

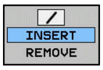
6.6 Skipping NC blocks

Insert a slash (/)

You can optionally hide NC blocks.

Proceed as follows to hide NC blocks in the **Programming** operating mode:

- ▶ Select the desired NC block

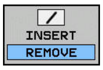


- ▶ Press the **INSERT** soft key
- > The control inserts a slash (/).

Delete the slash (/)

Proceed as follows to show NC blocks in the **Programming** operating mode again:

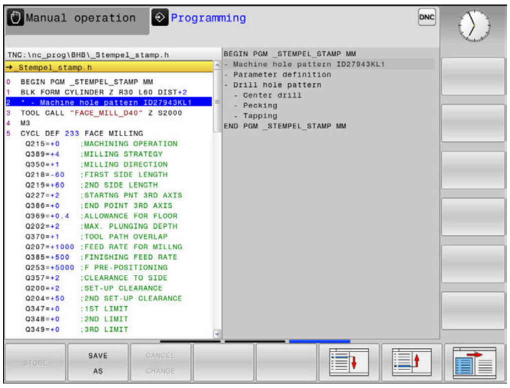
- ▶ Select the hidden NC block



- ▶ Press the **REMOVE** soft key
- > The control removes the slash (/).

6.7 Structuring NC programs

Definition and applications



The control enables you to comment NC programs in structuring blocks. Structuring blocks are texts with up to 252 characters and are used as comments or headlines for the subsequent program lines.

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

This function is particularly convenient if you want to change the NC program later. Structuring blocks can be inserted into the NC program at any point.

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

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

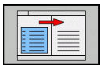
The **PROGRAM + SECTS** screen layout can be selected in the following operating modes:

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

Displaying the program structure window / Changing the active window

PROGRAM + SECTS

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



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

Inserting a structure block in the program window

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

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM-MING AIDS

- Press the **PROGRAMMING AIDS** soft key

INSERT SECTION

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

- Change the structuring depth (indenting) via soft key

> **i** The structure items can be indented only during editing.

Selecting blocks in the program structure window

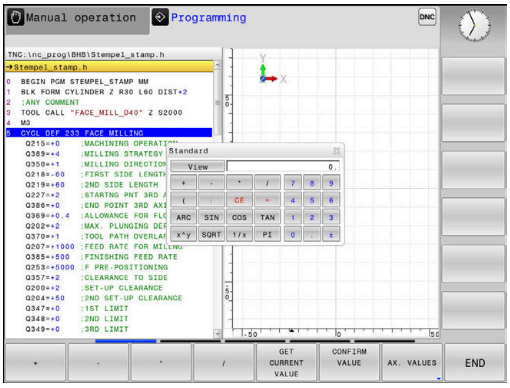
If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

6.8 Calculator

Operation

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

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



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

Calculate function	Shortcut (soft key)
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 NC program

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

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

Functions in the pocket calculator

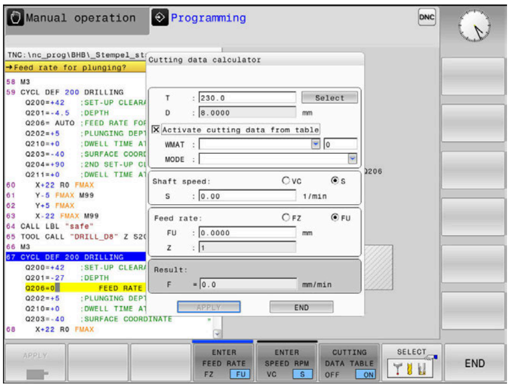
Soft key	Function
AX. VALUES	Load the nominal or reference value of the respective axis position into the calculator
GET CURRENT VALUE	Load the numerical value from the active input field into the calculator
CONFIRM VALUE	Load the numerical value from the calculator field into the active input field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

You can also move the calculator with the arrow keys of your alphabetic keyboard. If you have connected a mouse you can also position the calculator with this.

6.9 Cutting data calculator

Application

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



To open the cutting data calculator, press the **CUTTING DATA CALCULATOR** soft key.

The control shows the soft key if you

- press the **CALC** 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
- press the **F** soft key in the **Manual Operation** mode
- press the **S** soft key in the **Manual Operation** mode

Display modes of the cutting data calculator

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

Window for spindle speed calculation:

Abbrev.	Meaning
T:	Tool number
D:	Diameter of the tool
VC:	Cutting speed
S=	Result for spindle speed

If you open the speed calculator in a dialog where the tool is already defined, the speed calculator automatically applies the tool number and diameter. You only need to enter **VC** in the dialog field.

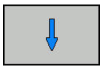
Window for feed rate calculation:

Abbrev.	Meaning
T:	Tool number
D:	Diameter of the tool
VC:	Cutting speed
S:	Spindle speed
Z:	Number of teeth
FZ:	Feed per tooth
FU:	Feed per revolution
F=	Result for feed rate

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

Functions of the cutting data calculator

You have the following possibilities depending on where you open the cutting data calculator:

Soft key	Function
APPLY	Transfer the value from the cutting data calculator into the NC program
CALCULATE FEEDRATE F SPEED S	Toggle between feed-rate calculation and spindle-speed calculation
ENTER FEED RATE FZ FU	Toggle between feed per tooth and feed per revolution
ENTER SPEED RPM VC S	Toggle between spindle speed and cutting speed
CUTTING DATA TABLE OFF ON	Activate or deactivate working with cutting data tables
SELECT	Select a tool from the tool table
	Move the cutting data calculator in the direction of the arrow
POCKET CALCULATOR	Switch to the calculator
INCH	Use inch values in the cutting data calculator
END	Close the cutting data calculator

Working with cutting data tables

Application

If you store tables for materials, cutting materials, and cutting data on the control, then the cutting data calculator can use the values in these tables.

Proceed as follows before working with automatic calculation of the spindle speed and feed rate:

- Enter the type of workpiece material in the table WMAT.tab
- Enter the type of cutting material in the file TMAT.tab
- Enter the combination of workpiece material and cutting material in a cutting data table
- Define the tool with the necessary values in the tool table
  - Tool radius
  - Number of teeth
  - Cutting material
  - Cutting data table

Workpiece material WMAT

Define the workpiece materials in the WMAT.tab table. You must save this table in the directory **TNC:\table**.

This table contains the column **WMAT** for the material and a column called **MAT_CLASS**; here you categorize the materials into material classes with the same cutting conditions, e.g. according to DIN EN 10027-2.

Enter the workpiece material as follows in the cutting data calculator:

- Select the cutting data calculator
- Select **Activate cutting data from table** in the pop-up window
- Select **WMAT** from the drop-down menu

TNC:\table\WMAT.TAB

NR	WMAT	MAT_CLASS
1		10
2	1.0038	10
3	1.0044	10
4	1.0114	10
5	1.0177	10
6	1.0143	10
7	St 37-2	10
8	St 37-3 N	10
9	X 14 CrMo S 17	20
10	1.1404	20
11	1.4305	20
12	V2A	21
13	1.4301	21
14	AlCu4PBMg	100
15	Aluminium	100
16	PTFE	200

Cutting material TMAT

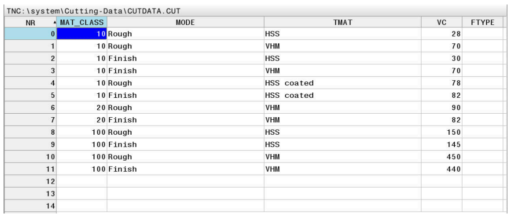
Cutting materials are defined in the TMAT.tab table. You must save this table in the directory **TNC:\table**.

You assign the cutting material in the **TMAT** column of the tool table. You can create columns with other names, such as **ALIAS1** and **ALIAS2** in order to enter alternative names for the same cutting material.

Cutting data table

Define the combinations of workpiece material and cutting material with the corresponding cutting data in a table with the file extension .CUT. You must save this table in the directory **TNC:\system\Cutting-Data**.

You assign the appropriate cutting data table in the **CUTDATA** column of the tool table.



TNC:\system\Cutting-Data\CUTDATA.CUT

NR	MAT_CLASS	MODE	TMAT	VC	FTYPE
0	10	Rough	HSS	28	
1	10	Rough	VHM	70	
2	10	Finish	HSS	30	
3	10	Finish	VHM	70	
4	10	Rough	HSS coated	78	
5	10	Finish	HSS coated	82	
6	20	Rough	VHM	90	
7	20	Finish	VHM	82	
8	100	Rough	HSS	150	
9	100	Finish	HSS	145	
10	100	Rough	VHM	450	
11	100	Finish	VHM	440	
12					
13					
14					

> Use this simplified table if you use tools that have only a single diameter, or if the diameter is not relevant to the feed rate, i.e. for indexable inserts.

The cutting data table contains the following columns:

- **MAT_CLASS**: Material class
- **MODE**: Machining mode, such as finishing
- **TMAT**: Cutting material
- **VC**: Cutting speed
- **FTYPE**: Type of feed rate **FZ** or **FU**
- **F**: Feed rate

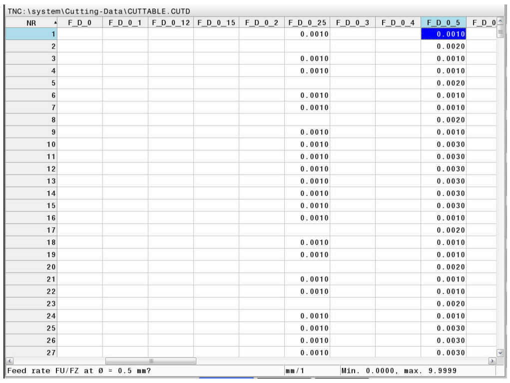
Diameter-dependent cutting data table

In many cases the diameter of the tool determines which cutting data you can use. Use the cutting data table with the file extension .CUTD for this purpose. You must save this table in the directory **TNC:\system\Cutting-Data**.

You assign the appropriate cutting data table in the **CUTDATA** column of the tool table.

The diameter-dependent cutting data table contains the following additional columns:

- **F_D_0**: Feed rate for Ø 0 mm
- **F_D_0_1**: Feed rate for Ø 0.1 mm
- **F_D_0_12**: Feed rate for Ø 0.12 mm
- ...



TNC:\system\Cutting-Data\CUTTABLE.CUTD

NR	F_D_0	F_D_0_1	F_D_0_12	F_D_0_15	F_D_0_2	F_D_0_25	F_D_0_3	F_D_0_4	F_D_0_5	F_D_0
1						0.0010			0.0010	
2									0.0020	
3						0.0010			0.0010	
4						0.0010			0.0010	
5									0.0020	
6						0.0010			0.0010	
7						0.0010			0.0010	
8									0.0020	
9						0.0010			0.0010	
10						0.0010			0.0030	
11						0.0010			0.0030	
12						0.0010			0.0030	
13						0.0010			0.0030	
14						0.0010			0.0030	
15						0.0010			0.0030	
16						0.0010			0.0010	
17									0.0020	
18						0.0010			0.0010	
19						0.0010			0.0010	
20									0.0020	
21						0.0010			0.0010	
22						0.0010			0.0010	
23									0.0020	
24						0.0010			0.0010	
25						0.0010			0.0030	
26						0.0010			0.0030	
27						0.0010			0.0030	

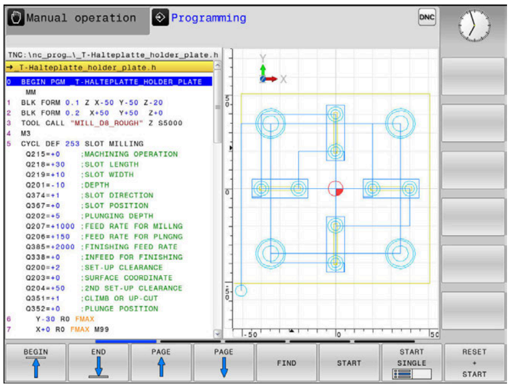
Feed rate FU/FZ at Ø = 0.5 mm? | mm/1 | Min. 0.0000, max. 9.9999

> You don't need to fill in all columns. If a tool diameter is between two defined columns, the control linearly interpolates the feed rate.

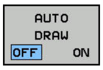
6.10 Programming graphics

Activating and deactivating programming graphics

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



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



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

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

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

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

The control uses various colors in the programming graphics:

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

Generating a graphic for an existing NC program

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

RESET + START

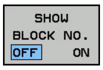
- Reset previously active tool data and generate graphics: Press the **RESET + START** soft key

Additional functions:

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

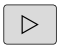
Block number display ON/OFF

- 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**

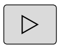
Erasing the graphic



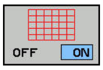
- Shift the soft-key row

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

Showing grid lines



- Shift the soft-key row



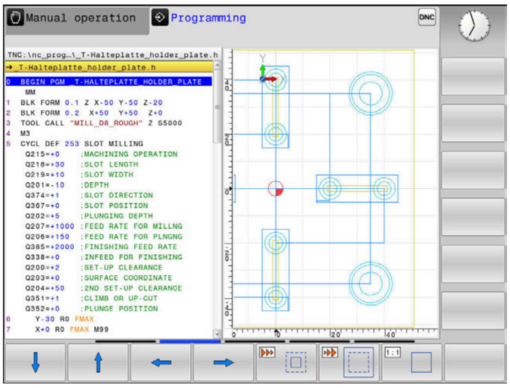
- Show grid lines: Press the **Show grid lines** soft key

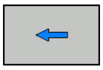
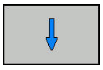
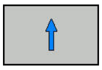
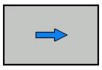
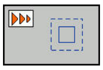
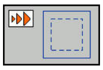
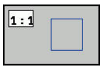
Magnification or reduction of details

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

6.11 Error messages

Display of errors

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

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

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

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

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

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

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

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

Opening the error window

ERR

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

Closing the error window

END

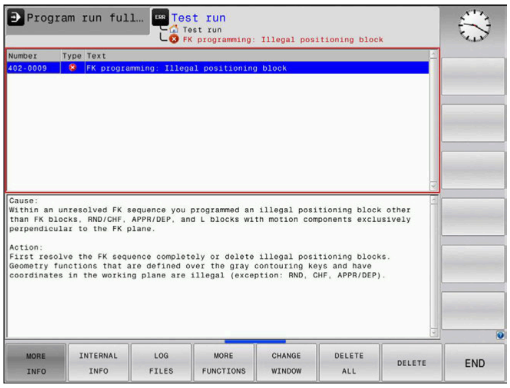
- ▶ Press the **END** soft key, or

ERR

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

Detailed error messages

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



- Open the error window

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

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 control opens a window with internal information about the error.
- Exit the details: Press the **INTERNAL INFO** soft key again

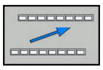
Soft key FILTER

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



- Exit the filter: Press the **GO BACK** soft key

Clearing errors

Clearing errors outside of the error window

CE

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

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

Clearing errors

- Open the error window

DELETE

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

DELETE ALL

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

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

Error log

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

- Open the error window.

LOG FILES

- Press the **LOG FILES** soft key

ERROR LOG

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

PREVIOUS FILE

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

CURRENT FILE

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

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

Keystroke log

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

LOG FILES
- Press the **LOG FILES** soft key

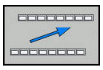
KEYSTROKE LOG
- Open the keystroke log file: Press the **KEYSTROKE LOG** soft key

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

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

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

Overview of the keys and soft keys for viewing the log

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

Informational texts

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

Saving service files

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

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

Saving service files

- Open the error window

LOG FILES

- Press the **LOG FILES** soft key

SAVE SERVICE FILES

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

OK

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

Calling the TNCguide help system

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

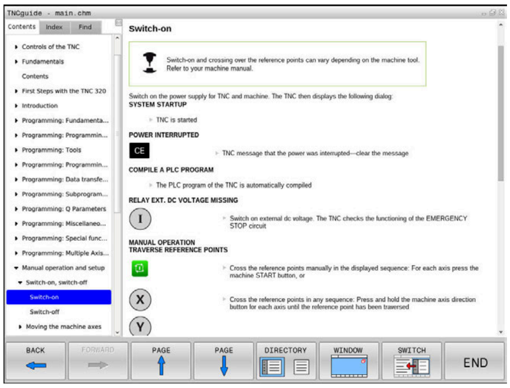
> Refer to your machine manual.
>
> If your machine manufacturer also provides a help system, the control shows an additional **Machine manufacturer (OEM)** soft key with which you can call this separate help system. There you will find further, more detailed information on the error message concerned.

6.12 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 149



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

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

The following user documentation is available in TNCguide:

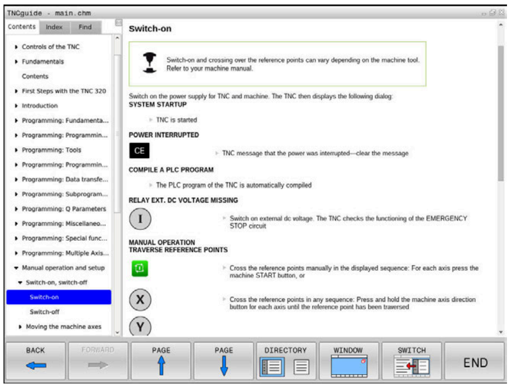
- Conversational Programming User's Manual (**BHBKlartext.chm**)
- User's Manual for Setup, Testing and Running NC Programs (**BHBoperate.chm**)
- List of All Error Messages **(errors.chm)**

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

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

Working with TNCguide

Calling TNCguide



There are several ways to start the TNCguide:

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

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

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

- ▶ Select the soft-key row containing the desired soft key
- ▶ Click with the mouse on the help symbol that the control displays just above the soft-key row
- > The mouse pointer turns into a question mark.
- ▶ Move the question mark to the soft key for which you want an explanation
- > The control opens the TNCguide. If there is no entry point for the selected soft key, then the control opens the book file **main.chm**. You can search for the desired explanation using full text search or by using the navigation.

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

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

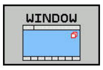
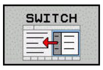
Navigating in the TNCguide

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

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

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

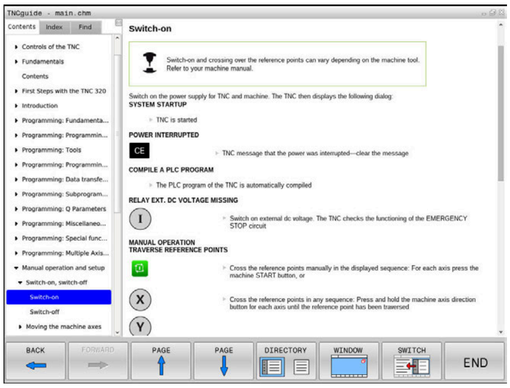
Soft key	Function
↑ ↓	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
→	■ If the table of contents at left is active: Open up the table of contents ■ If the text window at right is active: No function
←	■ If the table of contents at left is active: Close the table of contents ■ If the text window at right is active: No function
ENT	■ If the table of contents at left is active: Use the cursor key to show the selected page ■ If the text window at right is active: If the cursor is on a link, jump to the linked page
	■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the screen half at right ■ If the text window at right is active: Jump back to the window at left
 	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Jump to next link
BACK ←	Select the page last shown
FORWARD →	Page forward if you have used the **Select page last shown** function
PAGE ↑	Move up by one page
PAGE ↓	Move down by one page

Soft key	Function
	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 control window
	The focus is switched internally to the control application so that you can operate the control when the TNCguide is open. If the full screen is active, the control reduces the window size automatically before the change of focus
	Exit TNCguide

Subject index

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

The left side is active.



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

Alternative:

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

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

Full-text search

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

The left side is active.

- Select the **Find** tab
- Activate the **Find:** entry field
- Enter the search word
- Press the **ENT** key
- > The control lists all sources containing the word.
- Use the arrow keys to navigate to the desired source
- Press the **ENT** key to go to the selected source

> The full-text search only works for single words.
>
> If you activate the **Search only in titles** function, the control searches only through headings and ignores the body text. To activate the function, use the mouse or select it and then press the space bar to confirm.
>
> You can enter the search word only with an alphabetic keyboard connected via USB.

Downloading current help files

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

Navigate to the suitable help file as follows:

- TNC Controls
- Series, e.g. TNC 100
- Desired NC software number, e.g.TNC 128 (77184x-07)
- Select the desired language version from the **TNCguide online help** table
- Download the ZIP file
- Extract the ZIP file
- Move the extracted CHM files to the **TNC:\tncguide\en** directory or the respective language subdirectory on the control

When using **TNCremo** to transfer the CHM files to the control, select the binary mode for files with the **.chm** extension.

Language	TNC directory
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**
Chinese (traditional)	**TNC:\tncguide\zh-tw**
Slovenian	**TNC:\tncguide\sl**
Norwegian	**TNC:\tncguide\no**
Slovak	**TNC:\tncguide\sk**
Korean	**TNC:\tncguide\kr**
Turkish	**TNC:\tncguide\tr**
Romanian	**TNC:\tncguide\ro**

7

Miscellaneous Functions

7.1 Entering miscellaneous functions M

Fundamentals

With the control’s miscellaneous functions—also called M functions—you can affect:

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

You can enter up to four M (miscellaneous) functions at the end of a positioning block or in a separate NC block. The control displays the following dialog question: **Miscellaneous function M ?**

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

In the **Manual operation** and **Electronic handwheel** operating modes, the M functions are entered with the **M** soft key.

Effectiveness of miscellaneous functions

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

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

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

If multiple functions were programmed in a single NC block, the execution sequence is as follows:

- M functions taking effect at the start of the block are executed before those taking effect at the end of the block
- If all M functions are effective at the start or end of the block, execution takes place in the sequence as programmed

7.2 Miscellaneous functions for program run inspection, spindle and coolant

Overview

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

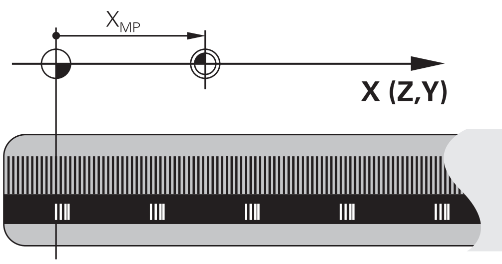
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope depends on machine parameter **resetAt** (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M6	Tool change Spindle STOP Program STOP			■
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

7.3 Miscellaneous functions for coordinate entries

Programming machine-referenced coordinates: M91/ M92

Scale datum

On the scale, a reference mark indicates the position of the scale datum.



Machine datum

The machine datum is required for the following tasks:

- Define the axis traverse limits (software limit switches)
- Approach machine-referenced positions (e.g. tool change positions)
- Set a workpiece preset

The distance in each axis from the scale datum to the machine datum is defined by the machine manufacturer in a machine parameter.

Standard behavior

The control references the coordinates to the workpiece datum.

Further information: User's Manual for Setup, Testing and Running NC Programs

Behavior with M91 – Machine datum

If you want the coordinates in a positioning block to be based on the machine datum, enter M91 into these NC blocks.

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

The coordinate values on the control screen reference the machine datum. Switch the display of coordinates in the status display to REF.

Further information: User's Manual for Setup, Testing and Running NC Programs

Behavior with M92 – Additional machine reference point

Refer to your machine manual.

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

For each axis, the machine tool builder defines the distance between the machine reference point and the machine datum.

If you want the coordinates in positioning blocks to be based on the machine preset, enter M92 into these NC blocks.

Radius compensation remains the same in blocks that are programmed with **M91** or **M92**. The tool length will **not** be taken into account.

Effect

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

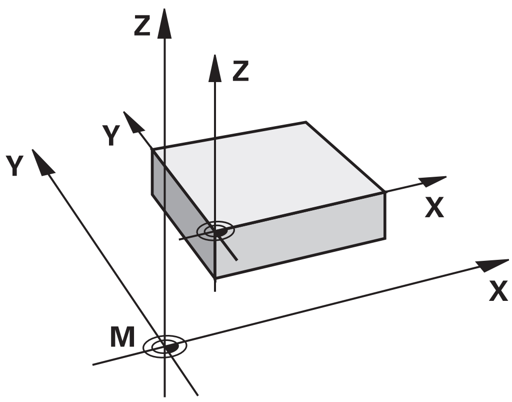
M91 and M92 take effect at the start of block.

Workpiece preset

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

If presetting is inhibited for all axes, the control no longer displays the **SET PRESET** soft key in the **Manual operation** mode.

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



M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the defined preset.

Further information: User's Manual for Setup, Testing and Running NC Programs

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

Standard behavior

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

Example:

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

Behavior with M94

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

If you entered a traverse limit or a software limit switch is active, **M94** is ineffective for the corresponding axis.

Example: Reduce the display of all active rotary axes

```
M94
```

Example: Reduce the display of the C axis

```
M94 C
```

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

```
C+180  FMAX M94
```

Effect

M94 is effective only in the NC block where it is programmed.

M94 becomes effective at the start of the block.

7.4 Miscellaneous functions for path behavior

Feed rate factor for plunging movements: M103

Standard behavior

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

Behavior with M103

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

FZMAX = FPROG x F%

Programming M103

If you program **M103** in a positioning block, the control continues the dialog by prompting you for the F factor.

Effect

M103 becomes effective at the start of the block.
Cancel **M103**: Program **M103** once again without a factor.

Feed rate in millimeters per spindle revolution: M136

Standard behavior

The control moves the tool at the feed rate F in mm/min programmed in the NC program

Behavior with M136

> In NC programs based on inch units, **M136** is not allowed in combination with the alternative **FU** feed rate.
>
> The spindle is not permitted to be controlled when M136 is active.

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

Effect

M136 becomes effective at the start of the block.

You can cancel **M136** by programming **M137**.

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

Standard behavior

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control moves the tool as defined in the NC program.

Behavior with M140

With **M140 MB** (move back), you can retract the tool from the contour by a programmable distance in the direction of the tool axis.

Input

If you enter **M140** in a positioning block, the control continues the dialog and prompts you for the path the tool should use for retracting from the contour. Enter the desired path that the tool should follow when retracting from the contour, or press the **MB MAX** soft key to move to the limit of the traverse range.

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

Effect

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

M140 becomes effective at the start of the block.

Example

NC block 250: Retract the tool by 50 mm from the contour

NC 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.

8

Subprograms and Program Section Repeats

8.1 Labeling subprograms and program section repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as necessary.

Label

The beginnings of subprograms and program section repeats in NC programs are marked by **(LBL)** labels.

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 NC program with the **LABEL SET** key. The number of label names you can enter is only limited by the internal memory.

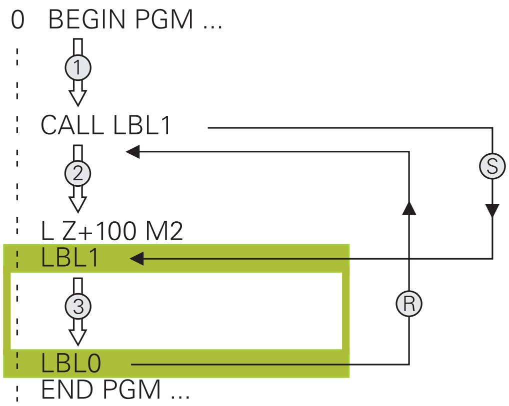
> Do not use a label number or label name more than once!

Label 0 (**LBL 0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

8.2 Subprograms

Operating sequence

1. The control executes the NC program up to the block in which a subprogram is called with **CALL LBL**
2. The subprogram is then executed until the subprogram end **LBL 0**
3. The control then resumes the NC program from the NC 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 NC block with M2 or M30
- If subprograms are located in the NC program before the NC block with M2 or M30, they will be executed at least once even if they are not called

Programming the subprogram

- To mark the beginning: Press the **LBL SET** key
- Enter the subprogram number. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the text
- Mark the end: Press the **LBL SET** key and enter the label number **0**

Calling a subprogram

LBL CALL

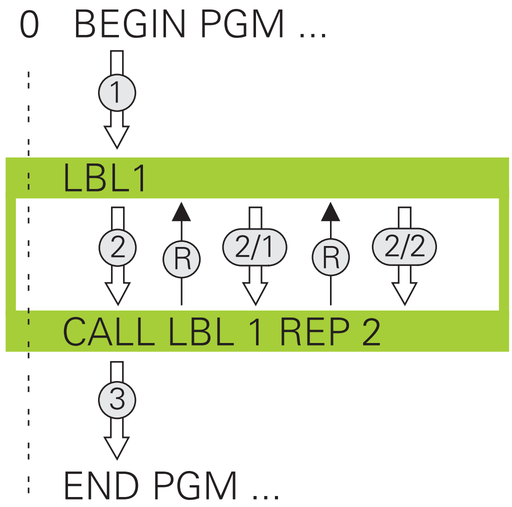
- Call a subprogram: Press the **LBL CALL** key
- Enter the subprogram number of the subprogram you wish to call. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- If you want to enter the number of a string parameter as target address, press the QS soft key
- > The control then jumps to the label name that is specified in the string parameter defined.
- Ignore repeats **REP** by pressing the **NO ENT** key. Repeat **REP** is used only for program section repeats

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

8.3 Program-section repeats

Label

The beginning of a program section repeat is marked by the label **LBL**. The end of a program section repeat is identified by **CALL LBL n REPn**.



Operating sequence

1. The control executes the NC program up to the end of the program section (**CALL LBL n REPn**)
2. Then the program section between the called LABEL and the label call **CALL LBL n REPn** is repeated the number of times entered after **REP**
3. The control then resumes the NC 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.

Programming a program section repeat

LBL SET

- To mark the beginning, press the **LBL SET** key and enter a LABEL NUMBER for the program section you wish to repeat. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the program section

Calling a program section repeat

LBL CALL

- Call a program section: Press the **LBL CALL** key
- Enter the program section number of the program section to be repeated. If you want to use a LABEL name, press the **LBL NAME** soft key to switch to text entry
- Enter the number of repeats **REP** and confirm with the **ENT** key.

8.4 Any desired NC program as subprogram

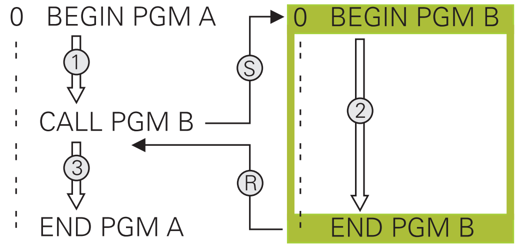
Overview of the soft keys

When you press the **PGM CALL** key, the control displays the following soft keys:

Soft key	Function
CALL PROGRAM	Call an NC program with **PGM CALL**
SELECT DATUM TABLE	Select a datum table with **SEL TABLE**
SELECT POINT TABLE	Select a point table with **SEL PATTERN**
SELECT PROGRAM	Select an NC program with **SEL PGM**
CALL SELECTED PROGRAM	Call the last selected file with **CALL SELECTED PGM**
SELECT CYCLE	Select any NC program with **SEL CYCLE** as a fixed cycle

Operating sequence

1. The control executes the NC program up to the block in which another NC program is called with **CALL PGM**.
2. Then the other NC program is run from beginning to end.
3. The control then resumes the calling NC program with the NC block behind the program call.



Programming notes

- The control does not require any labels to call any part program
- The called NC program must not contain any **CALL PGM** call into the calling NC program (an endless loop ensues)
- The called NC program must not contain the miscellaneous functions **M2** or **M30**. If you have defined subprograms with labels in the called NC program, you can then replace M2 or M30 with the **FN 9: If +0 EQU +0 GOTO LBL 99** jump function
- If you want to call a ISO program, enter the file type .I after the program name.
- You can also call an NC program with Cycle **12 PGM CALL**.
- You can also call any NC program with the function **Select the cycle** (**SEL CYCLE**).
- As a rule, Q parameters are effective globally with a **PGM CALL**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program.

Checking the called NC programs

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. If you do not specifically rescind the coordinate transformations in the called NC program, these transformations will also take effect in the calling NC program. Danger of collision during machining!

- Reset used coordinate transformations in the same NC program
- Check the machining sequence using a graphic simulation if required

The control checks the called NC programs:

- If the called NC program contains the miscellaneous functions **M2** or **M30**, then the control displays a warning. The control automatically clears the warning as soon as you select another NC program.
- The control checks the called NC programs to see whether they are complete before running them. If the **END PGM** NC block is missing, the control aborts with an error message.

Further information: User's Manual for Setup, Testing and Running NC Programs

Path information

If the NC program you want to call is located in the same directory as the NC program you are calling it from, then you only need to enter the program name.

If the called NC program is not located in the same directory as the NC program you are calling it from, you must enter the complete path, e.g. **TNC:\ZW35\HERE\PGM1.H**

Alternatively, you can program relative paths:

- Starting from the folder of the calling NC program one folder level up **..\PGM1.H**
- Starting from the folder of the calling NC program one folder level down **DOWN\PGM1.H**
- Starting from the folder of the calling NC program one folder level up and in one other folder **..\THERE\PGM3.H**

Calling an NC program as a subprogram

Calling a program with PGM CALL

The **PGM CALL** function calls any NC program as a subprogram. The control runs the called NC program from the position where it was called in the NC program.

Proceed as follows:

PGM CALL

- Press the **PGM CALL** key

CALL PROGRAM

- Press the **CALL PROGRAM** soft key
- > The control starts the dialog for defining the NC program to be called.
- Enter the path name with the keyboard

Alternative:

SELECT FILE

- Press the **SELECT FILE** soft key
- > The control displays a selection window in which you can select the NC program to be called.
- Press the **ENT** key

Call with SEL PGM and CALL SELECTED PGM

Use the function **SEL PGM** to select any NC program as a subprogram and call it at another position in the NC program. The control runs the called NC program from the position where you called it with **CALL SELECTED PGM** in the NC program.

The **SEL PGM** function is also permitted with string parameters, so that you can dynamically control program calls.

To select the NC program, proceed as follows:

PGM CALL

- Press the **PGM CALL** key

SELECT PROGRAM

- Press the **SELECT PROGRAM** soft key
- > The control starts the dialog for defining the NC program to be called.

SELECT FILE

- Press the **SELECT FILE** soft key
- > The control displays a selection window in which you can select the NC program to be called.
- Press the **ENT** key

To call the selected NC program, proceed as follows:

PGM CALL

- Press the **PGM CALL** key

CALL SELECTED PROGRAM

- Press the **CALL SELECTED PROGRAM** soft key
- > The control uses **CALL SELECTED PGM** to call the NC program that was selected last.

> If an NC program that was called using **CALL SELECTED PGM** is missing, then the control interrupts the execution or simulation with an error message. In order to avoid undesired interruptions during program run, you can use the function **FN 18** (**ID10 NR110** and **NR111**) to check all paths at the beginning of the program.
> **Further information:** "FN 18: SYSREAD – Reading system data", Page 208

8.5 Nesting

Types of nesting

- Subprogram calls in subprograms
- Program-section repeats within a program-section repeat
- Subprogram calls in program section repeats
- Program-section repeats in subprograms

Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

- Maximum nesting depth for subprograms: 19
- Maximum nesting depth for main program calls: 19, where a **CYCL CALL** acts like a main program call
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example

0 BEGIN PGM UPGMS MM	
...	
17 CALL LBL "UP1"	Call the subprogram marked with LBL SP1
...	
35 Z+100 R0 FMAX M2	Last program block of the main program with M2
36 LBL "UP1"	Beginning of subprogram SP1
...	
39 CALL LBL 2	Call the subprogram marked with LBL 2
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Beginning of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM SUBPGMS MM	

Program execution

1. Main program UPGMS is executed up to NC block 17
2. Subprogram UP1 is called, and executed up to NC block 39
3. Subprogram 2 is called, and executed up to NC block 62. End of subprogram 2 and return jump to the subprogram from which it was called.
4. Subprogram UP1 is called, and executed from NC block 40 up to NC block 45. End of subprogram 1 and return jump to the main program UPGMS.
5. Main program UPGMS is executed from NC block 18 up to NC block 35. Return jump to NC block 1 and end of program

Repeating program section repeats

Example

0 BEGIN PGM REPS MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2	Program section call with two repeats
...	
35 CALL LBL 1 REP 1	The program section between this NC block and LBL 1
...	(NC block 15) is repeated once
50 END PGM REPS MM	

Program execution

1 Main program REPS is executed up to NC block 27
2 The program section between NC block 27 and NC block 20 is repeated twice
3 Main program REPS is executed from NC block 28 up to NC block 35
4 The program section between NC block 35 and NC block 15 is repeated once (including the program section repeat between NC block 20 and NC block 27)
5 Main program REPS is executed from NC block 36 up to NC block 50. Return jump to NC block 1 and end of program

Repeating a subprogram

Example

0 BEGIN PGM UPGREP MM	
...	
10 LBL 1	Beginning of program section repeat 1
11 CALL LBL 2	Subprogram call
12 CALL LBL 1 REP 2	Program section call with two repeats
...	
19 Z+100 R0 FMAX M2	Last NC 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

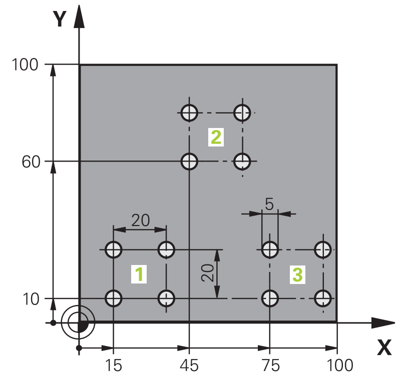
1 Main program UPGREP is executed up to NC block 11
2 Subprogram 2 is called and executed.
3 The program section between NC block 12 and NC block 10 is repeated twice. This means that subprogram 2 is repeated twice
4 Main program UPGREP is executed from NC block 13 up to NC block 19. Return jump to NC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

8.6 Programming examples

Example: Groups of holes

Program run:

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1) in the main program
- Program the group of holes only once in subprogram 1



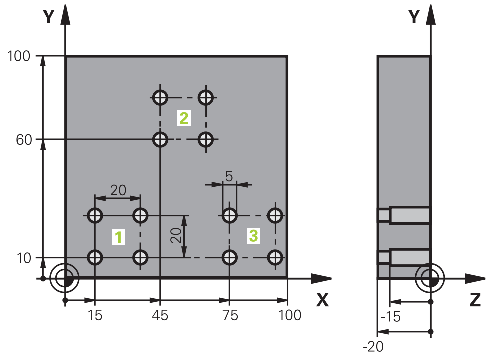
0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3000	Tool call
4 Z+250 R0 FMAX M3	
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q210=+0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DWELL TIME AT DEPTH	
Q395=+0 ;DEPTH REFERENCE	
6 CYCL DEF 7.0 DATUM SHIFT	Datum shift
7 CYCL DEF 7.1 X+15	
8 CYCL DEF 7.2 Y+10	
9 CALL LBL 1	
10 CYCL DEF 7.0 DATUM SHIFT	Datum shift
11 CYCL DEF 7.1 X+75	
12 CYCL DEF 7.2 Y+10	
13 CALL LBL 1	
14 CYCL DEF 7.0 DATUM SHIFT	Datum shift
15 CYCL DEF 7.1 X+45	
16 CYCL DEF 7.2 Y+60	
17 CALL LBL 1	
18 CYCL DEF 7.0 DATUM SHIFT	
19 CYCL DEF 7.1 X+0	

20 CYCL DEF 7.2 Y+0	
21 Z+100 R0 FMAX M30	
22 LBL 1	
23 X+0 R0 FMAX	
24 Y+0 R0 FMAX M99	Move to 1st hole, call cycle
25 X+20 R0 FMAX M99	Move to 2nd hole, call cycle
26 Y+20 R0 FMAX M99	Move to 3rd hole, call cycle
27 X-20 R0 FMAX M99	Move to 4th hole, call cycle
28 LBL 0	
29 END PGM SP2 MM	

Example: Group of holes with several tools

Program run:

- Program the fixed cycles in the main program
- Call the complete hole pattern (subprogram 1) in the main program
- Approach the groups of holes (subprogram 2) in subprogram 1
- Program the group of holes only once in subprogram 2



0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Centering drill tool call
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q201=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=3 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 CALL LBL 1	Call subprogram 1 for the entire hole pattern
7 Z+250 R0 FMAX M6	Tool change
8 TOOL CALL 2 Z S4000	Drill tool call
9 FN 0: Q201 = -25	New depth for drilling
10 FN 0: Q202 = +5	New plunging depth for drilling
11 CALL LBL 1	Call subprogram 1 for the entire hole pattern
12 Z+250 R0 FMAX M6	Tool change
13 TOOL CALL 3 Z S500	Reamer tool call

14 CYCL DEF 201 REAMING	Cycle definition: REAMING
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG.	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
15 CALL LBL 1	Call subprogram 1 for the entire hole pattern
16 Z+250 R0 FMAX M2	End of main program
17 LBL 1	Beginning of subprogram 1: Entire hole pattern
18 X+15 R0 FMAX M3	Move to starting point X for hole group 1
19 Y+10 R0 FMAX M3	Move to starting point Y for hole group 1
20 CALL LBL 2	Call subprogram 2 for the hole group
21 X+45 R0 FMAX	Move to starting point X for hole group 2
22 Y+60 R0 FMAX	Move to starting point Y for hole group 2
23 CALL LBL 2	Call subprogram 2 for the hole group
24 X+75 R0 FMAX	Move to starting point X for hole group 3
25 Y+10 R0 FMAX	Move to starting point Y for hole group 3
26 CALL LBL 2	Call subprogram 2 for the hole group
27 LBL 0	End of subprogram 1
28 LBL 2	Beginning of subprogram 2: Group of holes
29 CYCL CALL	1st hole with active fixed cycle
30 IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
31 IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
32 IX-20 R0 FMAX M99	Move to 4th hole, call cycle
33 LBL 0	End of subprogram 2
34 END PGM UP2 MM	

9

Programming Q Parameters

9.1 Principle and overview of functions

With Q parameters you can program entire families of parts in a single NC program by programming variable Q parameters instead of fixed numerical values.

Use Q parameters for e.g.:

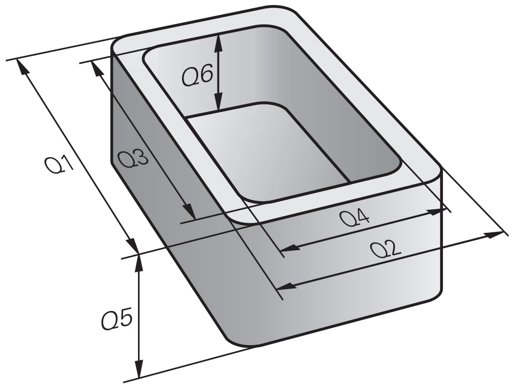
- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

With Q parameters you can also:

- Program contours that are defined through mathematical functions
- Make execution of machining steps depend on certain logical conditions

Q parameters are always identified with letters and numbers. The letters determine the type of Q parameter and the numbers the Q parameter range.

For more information, see the table below:



Q parameter type	Q parameter range	Meaning
Q parameters:		**Parameters affect all NC programs in the control's memory**
	0 to 99	Parameters for the **user**, if there are no overlaps with the HEIDENHAIN-SL cycles
	100 to 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 to 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 to 1399	Parameters preferentially used with manufacturer cycles if values are returned to the user program
	1400 to 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 to 1999	Parameters for **users**
QL parameters:		**Parameters only effective locally within an NC program**
	0 to 499	Parameters for **users**
QR parameters:		**Parameters permanently affect all NC programs in the control's memory, including after a power interruption**
	0 to 99	Parameters for **users**
	100 to 199	Parameters for HEIDENHAIN functions (e.g., cycles)
	200 to 499	Parameters for the machine tool builder (e.g., cycles)

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

Q parameter type	Q parameter range	Meaning
QS parameters:		**Parameters affect all NC programs in the control's** memory
	0 to 99	Parameters for the **user**, where no overlaps with the HEIDENHAIN SL cycles are present
	100 to 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 to 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 to 1399	Parameters preferentially used with manufacturer cycles if values are returned to the user program
	1400 to 1599	Parameters primarily used as input parameters for manufacturer cycles
	1600 to 1999	Parameters for **users**

NOTICE

Danger of collision!

HEIDENHAIN cycles, manufacturer cycles and third-party functions use Q parameters. You can also program Q parameters within NC programs. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.
- Check the machining sequence using a graphic simulation

Programming notes

You can mix Q parameters and numerical values within an NC program.

Q parameters can be assigned numerical values between -999 999 999 and +999 999 999. The input range is limited to 16 digits, of which 9 may be before the decimal point. Internally the control calculates numbers up to a value of 10^{10}.

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

> The control automatically assigns some Q and QS parameters the same data, e.g., the Q parameter **Q108** is automatically assigned the current tool radius.
>
> **Further information:** "Preassigned Q parameters", Page 248
>
> The control saves numerical values internally in a binary number format (standard IEEE 754). Due to the standardized format used, the control does not represent some decimal numbers with a binary number that is 100% exact (round-off error). If you use calculated Q parameter contents for jump commands or positioning moves, then you must take this fact into consideration.

You can reset Q parameters to the status **Undefined**. If a position is programmed with a Q parameter that is undefined, the control ignores this movement.

Calling Q parameter functions

When you are writing an NC program, press the **Q** key (in the numeric keypad for numerical input and axis selection, below the **+/-** key). The control then displays the following soft keys:

Soft key	Function group	Page
BASIC ARITHM.	Basic arithmetic (assign, add, subtract, multiply, divide, square root)	187
TRIGO-NOMETRY	Trigonometric functions	190
CIRCLE CALCU-LATION	Function for calculating circles	191
JUMP	If/then conditions, jumps	192
DIVERSE FUNCTION	Other functions	196
FORMULA	Entering formulas directly	231

If you define or assign a Q parameter, then the control shows the **Q**, **QL** and **QR** soft keys. You can use these soft keys to select the desired parameter type. Then you define the parameter number.

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

9.2 Part families—Q parameters in place of numerical values

Application

The Q parameter function **FN 0: ASSIGN** assigns numerical values to Q parameters. This enables you to use variables in the NC program instead of fixed numerical values.

Example

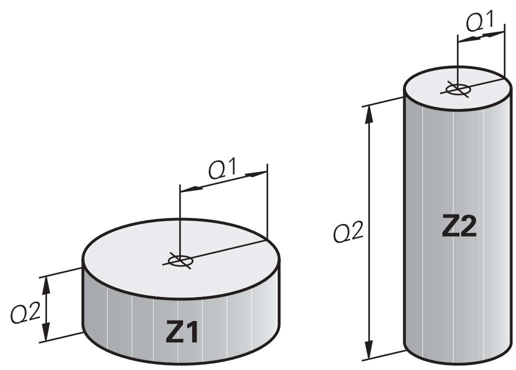
15 FN 0: Q10=25	Assign
...	Q10 is assigned the value 25
25 X +Q10	Means X +25

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example: Cylinder with Q parameters

Cylinder radius:	R = Q1
Cylinder height:	H = Q2
Cylinder Z1:	Q1 = +30 Q2 = +10
Cylinder Z2:	Q1 = +10 Q2 = +50



9.3 Describing contours with mathematical functions

Application

The Q parameters listed below enable you to program basic mathematical functions in a NC program:

- Select a Q parameter function: Press the **Q** key (in the numerical keypad on the right). The Q parameter functions are displayed in a soft key row
- To select the basic mathematical functions, press the **BASIC ARITHM...** soft key.
- > The control then displays the following soft keys:

Overview

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

You can enter the following to the right of the **=** sign:

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

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

Programming fundamental operations

ASSIGN

Example

```
16 FN 0: Q5 = +10
17 FN 3: Q12 = +Q5 * +7
```

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

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

FN0 X = Y
- To select the ASSIGN Q parameter function: Press the **FN 0 X = Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT
- Enter **5** (the number of the Q parameter) and confirm with the **ENT** key

FIRST VALUE / PARAMETER?

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

MULTIPLICATION

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

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

FN3 X * Y
- To select the MULTIPLICATION Q parameter function, press the **FN 3 X * Y** soft key

PARAMETER NUMBER FOR RESULT?

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

FIRST VALUE / PARAMETER?

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

SECOND VALUE / PARAMETER?

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

Resetting Q parameters

Example

```
16 FN 0: Q5 SET UNDEFINED
17 FN 0:  Q1 = Q5
```

Q

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

BASIC ARITHM.

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

FN0 X = Y

- To select the ASSIGN Q parameter function: Press the **FN 0 X = Y** soft key

PARAMETER NUMBER FOR RESULT?

ENT

- Enter **5** (the number of the Q parameter) and confirm with the **ENT** key

1. VALUE OR PARAMETER?

SET UNDEFINED

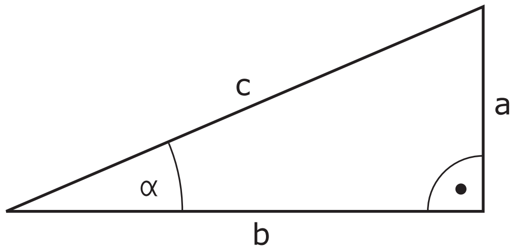
- Press **SET UNDEFINED**

The **FN 0** function also supports transfer of the value **Undefined**. If you wish to transfer the undefined Q parameter without **FN 0**, the control shows the error message **Invalid value**.

9.4 Trigonometric functions

Definitions

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



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

The control can find the angle from the tangent:
$\alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

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

Programming trigonometric functions

Press the **TRIGONOMETRY** soft key to call the trigonometric functions. The control then displays the soft keys listed in the table below:

Soft key	Function
FN6 SIN(X)	**FN 6**: SINUS e. g., **FN 6: Q20 = SIN-Q5** Calculate and assign the sine of an angle in degrees (°)
FN7 COS(X)	**FN 7**: COSINE e. g., **FN 7: Q21 = COS-Q5** Calculate and assign the cosine of an angle in degrees (°)
FN8 X LEN Y	**FN 8**: ROOT SUM OF SQUARES e. g., **FN 8: Q10 = +5 LEN +4** Calculate and assign lengths from two values
FN13 X ANG Y	**FN 13**: ANGLE e. g., **FN 13: Q20 = +25 ANG-Q1** Calculate and assign an angle with the arc tangent from the opposite and adjacent sides or with the sine and cosine of the angle (0 < angle < 360°)

9.5 Calculation of circles

Application

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

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

Soft key	Function
FN23 3 POINTS OF CIRCLE	FN 23: Determining the CIRCLE DATA from three points e. g., **FN 23: Q20 = CDATA Q30**

The coordinate pairs of three points on a circle must be saved in Q30 and the following five parameters—in this case, up to Q35.

The control then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.

Soft key	Function
FN24 4 POINTS OF CIRCLE	FN 24: Determining the CIRCLE DATA from four points e. g., **FN 24: Q20 = CDATA Q30**

The coordinate pairs of four points on a circle must be saved in Q30 and the following seven parameters—in this case, up to Q37.

The control then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.

> Note that **FN 23** and **FN 24** automatically overwrite the resulting parameter and the two following parameters.

9.6 If-then decisions with Q parameters

Application

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

Further information: "Labeling subprograms and program section repeats", Page 162

If it is not fulfilled, the control continues with the next NC block.

To call another NC program as a subprogram, enter a **PGM CALL** program call after the block with the label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

FN 9: IF+10 EQU+10 GOTO LBL1

Abbreviations used:

IF	:	If
EQU	:	Equal to
NE	:	Not equal to
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to
UNDEFINED	:	Undefined
DEFINED	:	Defined

Programming if-then decisions

Possibilities for jump inputs

The following inputs are possible for the condition **IF**:

- Numbers
- Texts
- Q, QL, QR
- **QS** (string parameter)

You have three possibilities for entering the jump address **GOTO**:

- **LBL NAME**
- **LBL NUMBER**
- **QS**

Press the **JUMP** soft key to call the if-then conditions. The control then displays the following soft keys:

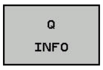
Soft key	Function
FN9 IF X EQ Y GOTO EQU	**FN 9**: IF EQUAL, JUMP e. g. **FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"** If both values or parameters are equal, jump to specified label
FN9 IF X EQ Y GOTO IS UNDEFINED	**FN 9**: IF UNDEFINED, JUMP e. g., **FN 9: IF +Q1 IS UNDEFINED GOTO LBL “UPCAN25“** If the specified parameter is undefined, then a jump is made to the specified label
FN9 IF X EQ Y GOTO IS DEFINED	**FN 9**: IF DEFINED, JUMP e. g., **FN 9: IF +Q1 IS DEFINED GOTO LBL “UPCAN25“** If the specified parameter is defined, then a jump is made to the specified label
FN10 IF X NE Y GOTO	**FN 10**: IF UNEQUAL, JUMP e. g.**FN 10: IF +10 NE -Q5 GOTO LBL 10** If both values or parameters are unequal, jump to specified label
FN11 IF X GT Y GOTO	**FN 11**: IF GREATER, JUMP g. g.**FN 11: IF+Q1 GT+10 GOTO LBL QS5** If the first value or parameter is greater than the second value or parameter, jump to specified label
FN12 IF X LT Y GOTO	**FN 12**: IF LESS, JUMP e. g. **FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME"** If the first value or parameter is smaller than the second value or parameter, jump to specified label

9.7 Checking and changing Q parameters

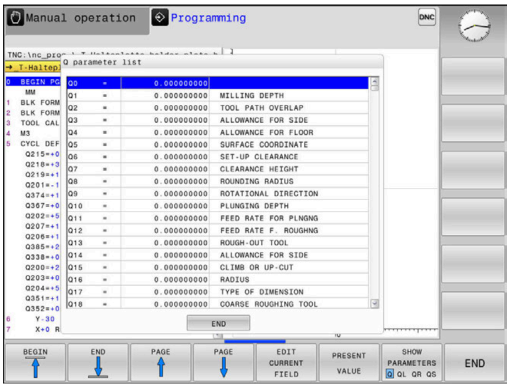
Procedure

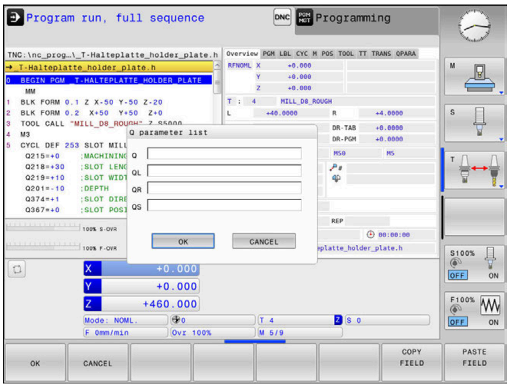
You can check Q parameters in all operating modes, and also edit them.

- If you are in a program run, interrupt it if required (e.g. by pressing the **NC 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 control lists all of the parameters and their corresponding current values.
- Use the arrow keys or the **GOTO** key to select the desired parameter.
- If you would like to change the value, press the **EDIT CURRENT FIELD** soft key. Enter a new value and confirm with the **ENT** key
- To leave the value unchanged, press the **PRESENT VALUE** soft key or close the dialog with the **END** key





All of the parameters with displayed comments are used by the control within cycles or as transfer parameters.

If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The control then displays the specific parameter type. The functions previously described also apply.

You can have Q parameters also displayed in the additional status display in all operating modes (except **Programming** mode).

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

PROGRAM + STATUS

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

STATUS OF Q PARAM.

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

Q PARAMETER LIST

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

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

9.8 Additional functions

Overview

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

Soft key	Function	Page
FN14 ERROR=	**FN 14: ERROR** Display error messages	197
FN16 F-PRINT	**FN 16: F-PRINT** Formatted output of texts or Q parameter values	201
FN18 SYS-DATUM READ	**FN 18: SYSREAD** Read system data	208
FN19 PLC=	**FN 19: PLC** Transfer values to the PLC	208
FN20 WAIT FOR	**FN 20: WAIT FOR** NC and PLC synchronization	209
FN26 OPEN TABLE	**FN 26: TABOPEN** Open a freely definable table	260
FN27 WRITE TO TABLE	**FN 27: TABWRITE** Write to a freely definable table	260
FN28 READ FROM TABLE	**FN 28: TABREAD** Read from a freely definable table	261
FN29 PLC LIST=	**FN 29: PLC** Transfer up to eight values to the PLC	209
FN37 EXPORT	**FN 37: EXPORT** Export local Q parameters or QS parameters into a calling NC program	210
FN38 SEND	**FN 38: SEND** Send information from the NC program	210

FN 14: ERROR: Displaying error messages

With the **FN 14: ERROR** error function, you can output error messages under program control. The messages are predefined by the machine tool builder or by HEIDENHAIN. If the control encounters an NC block with **FN 14: ERROR** during program run, it will interrupt the run and display an error message. You must then restart the NC program.

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

Example

The control is intended to display a message if the spindle is not switched on.

```
180 FN 14: ERROR = 1000
```

Error message predefined by HEIDENHAIN

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRROR IMAGE not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Input value incorrect
1012	Incorrect sign
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory input
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined

Error number	Text
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive nesting
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Q218 must be greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2

Error number	Text
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal to 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter Q351 unequal to 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as negative
1078	Q303 in meas. cycle undefined!
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory meas. points
1082	Incorrect clearance height
1083	Contradictory plunge type
1084	This fixed cycle not allowed
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not allowed
1090	Enter an infeed not equal to 0
1091	Switchover of Q399 not allowed
1092	Tool not defined
1093	Tool number not permitted

Error number	Text
1094	Tool name not permitted
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed
1100	Kinematic access not possible
1101	Meas. pos. not in traverse range
1102	Preset compensation not possible
1103	Tool radius too large
1104	Plunging type is not possible
1105	Plunge angle incorrectly defined
1106	Angular length is undefined
1107	Slot width is too large
1108	Scaling factors not equal
1109	Tool data inconsistent

FN 16: F-PRINT – Formatted output of text and Q parameter values

Basics

With the function **FN 16: F-PRINT**, you can save Q parameter values and output formatted texts (e.g. in order to save measurement reports).

You can output the values as follows:

- Save them to a file on the control
- Display them on the screen in a pop-up window
- Save them to an external file
- Print them using a connected printer

Procedure

Proceed as follows in order to output Q-parameter values and texts:

- Create a text file that defines the output format and contents
- In the NC program, use the function **FN 16: F-PRINT** in order to output the log

If you output the values to a file, the maximum size if the output file will be 20 KB.

In machine parameters **fn16DefaultPath** (no. 102202) and **fn16DefaultPathSim** (no. 102203) you can define a default path for outputting log files.

Creating a text file

To output the formatted texts and Q parameter values, use the control's text editor to create a text file. Define the format and Q parameters to be output in this file.

Proceed as follows:

PGM MGT

- Press the **PGM MGT** key

NEW FILE

- Press the **NEW FILE** soft key
- Create a file with the extension **.A**

Available functions

Use the following formatting functions for creating a text file:

Special characters	Function
"..........."	Define output format for texts and variables between the quotation marks
%F	Format for Q parameters, QL, and QR: ■ Define %: format ■ F: Floating (decimal number), format for Q, QL, QR
9.3	Format for Q parameters, QL, and QR: ■ Total of 9 characters, including decimal separator ■ Of these, 3 are decimal places
%S	Format for text variable QS
%RS	Format for text variable QS Assumes the subsequent without any changes or formatting
%D or **%I**	Format for integer
,	Separation character between output format and parameter
;	End of block character
*****	Beginning of a comment line Comments are not shown in the log
\n	Line break
+	Q parameter value, right-aligned
-	Q parameter value, left-aligned

Example

Input	Meaning
"X1 = %+9.3F", Q31;	Format for Q parameter: ■ **"X1 =**: The text **X1 =** is output ■ **%**: Specify the format ■ **+**: Number right-aligned ■ **9.3**: Total of 9 characters; 3 of them are decimal places ■ **F**: Floating (decimal number) ■ **, Q31**: Output the value from Q31 ■ **;**: End of block

The following functions allow you to include the following additional information in the protocol log file:

Keyword	Function
CALL_PATH	Gives the path for the NC program where you will find the FN 16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with FN 16. Example: M_CLOSE;
M_APPEND	Upon renewed output, appends the log to the existing log. Example: M_APPEND;
M_APPEND_MAX	Upon renewed output, appends the log to the existing log until the maximum specified file size in kilobytes is exceeded. Example: M_APPEND_MAX20;
M_TRUNCATE	Overwrites the log upon renewed output. Example: M_TRUNCATE;
L_ENGLISH	Outputs the text only if English is set as dialog language
L_GERMAN	Outputs the text only if German is set as dialog 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

Keyword	Function
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_NORWEGIAN	Outputs text only for Norwegian conversational language
L_ROMANIAN	Outputs text only for Romanian conversational language
L_SLOVAK	Outputs text only for Slovakian conversational language
L_TURKISH	Outputs text only for Turkish conversational language
L_ALL	Display text independently of the conversational language
HOUR	Number of hours from the real-time clock
MIN	Number of minutes from the real-time clock
SEC	Number of seconds from the real-time clock
DAY	Day from the real-time clock
MONTH	Month as a number from the real-time clock
STR_MONTH	Month as a string abbreviation from the real-time clock
YEAR2	Two-digit year from the real-time clock
YEAR4	Four-digit year from the real-time clock

Example

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;
L_GERMAN;
"Werkzeuglänge beachten";
L_ENGLISH;
"Remember the tool length";
```

Activating FN 16 output in an NC program

Within the **FN 16** you specify the output file that contains the texts to be output.

The control generates the output file:

- at the end of the program (**END PGM**),
- if a program is canceled (**NC STOP** key)
- as a result of the command **M_CLOSE**

Enter the path of the source and the path of the output file in the FN 16 function .

Proceed as follows:

Q

- Press the **Q** key.

DIVERSE FUNCTION

- Press the **DIVERSE FUNCTION** soft key

FN16 F-PRINT

- Press the **FN16 F-PRINT** soft key

SELECT FILE

- Press the **SELECT FILE** soft key
- Select the source, i.e. the text file in which the output file is defined

ENT

- Confirm with the **ENT** key
- Enter the output path.

Path entries in the FN 16 function

If you enter only the file name as the path for the log file, the control saves the log file in the directory in which the NC program with the **FN 16** function is located.

Program relative paths as an alternative to complete paths:

- Starting from the folder of the calling file one folder level down **FN 16: F-PRINT MASKE\MASKE1.A/ PROT\PROT1.TXT**
- Starting from the folder of calling file one folder level up and in another folder **FN 16: F-PRINT ..\MASKE\MASKE1.A/ ..\PROT1.TXT**

> Operating and programming notes:
> - If you output the same file more than once in the NC program, the control appends the current output to the end of the contents already output within the target file.
> - In the **FN 16** block, program the format file and the log file, each with the extension for the file type.
> - The file name extension of the log file determines the file format of the output (e.g., TXT, .A, .XLS, .HTML).
> - If you use **FN 16**, then no UTF-8 encoding is permitted for the file.
> - Use **FN 18** to receive much information that is relevant and interesting in log files, such as the number of the touch-probe cycle last used.
>   **Further information:** "FN 18: SYSREAD – Reading system data", Page 208

Enter the source or the target with parameters

You can enter the source file and the output file as Q parameters or as QS parameters. For this purpose you previously define the desired parameter in the NC program.

Further information: "Assign string parameters", Page 236

Enter Q parameters in the **FN 16** function with the following syntax so that the control can detect the Q parameters:

Input	Function
:'QS1'	Set QS parameters with preceding colon and between single quotation marks
:'QL3'.txt	Specify additional file name extension for the target file if required

> If you want to output a path with a QS parameter to a log file, then use the function **%RS**. This ensures that the control does not interpret the special characters as formatting characters.

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/ TNC:\PROT1.TXT
```

The control 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

Remember the tool length

Displaying messages on the control screen

You can also use the function **FN 16: F-PRINT** to display any messages from the NC program in a pop-up window on the control screen. This makes it easy to display explanatory texts, including long texts, at any point in the NC program in a way that the user has to react to them. You can also display Q-parameter contents if the protocol description file contains such instructions.

For the message to appear on the control screen, you need only enter **SCREEN:** as the output path.

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.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.

> If you want to overwrite the previous pop-up window, program the function **M_CLOSE** or **M_TRUNCATE**.

Close the pop-up window

You can close the pop-up window in the following ways:

- Press the **CE** key
- Controlled by the program with the output path **sclr:**

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/SCLR:
```

Exporting messages

With the **FN 16** function you can also store log files externally.

To do so you must enter the target path in the **FN 16** function.

Example

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

> If you output the same file more than once in the NC program, the control appends the current output to the end of the contents already output within the target file.

Printing messages

You can also use the function **FN 16: F-PRINT** to print any messages on a connected printer.

Further information: User's Manual for Setup, Testing and Running NC Programs

In order for the messages to be sent to the printer, you must enter **Printer:** as the name of the log file and then enter the corresponding file name.

The control saves the file in the **PRINTER:** path until the file is printed.

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/PRINTER:\DRUCK1
```

FN 18: SYSREAD – Reading system data

With the **FN 18: SYSREAD** function, you can read system data and save them to Q parameters. The selection of the system datum occurs via a group number (ID no.), a system data number, and, if necessary, an index.

> The read values of the function **FN 18: SYSREAD** are always output by the control in **metric** units regardless of the NC program's unit of measure.

Further information: "System data", Page 436

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

```
55 FN 18: SYSREAD Q25 = ID210 NR4 IDX3
```

FN 19: PLC – Transfer values to the PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

The **FN 19: PLC** function transfers up to two numerical values or Q parameters to the PLC.

FN 20: WAIT FOR – NC and PLC synchronization

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

With the **FN 20: WAIT FOR** function you can synchronize the NC and PLC during a program run. The NC stops machining until the condition that you have programmed in the **FN 20: WAIT FOR** block is fulfilled.

SYNC is used whenever you read, for example, system data via **FN 18: SYSREAD** that require synchronization with real time. The control stops the look-ahead calculation and executes the following NC block only when the NC program has actually reached that NC 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 – Transferring values to the PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

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

FN 37: EXPORT

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. The FN function provides HEIDENHAIN as well as your machine tool builder and suppliers the ability to communicate with the PLC from an NC program. It is not recommended that the machine operator or NC programmer use this. There is risk of collision during the execution of the function and during the subsequent processing!

- Only use the function in consultation with HEIDENHAIN, the machine tool builder, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.

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

FN 38: SEND – Send information from NC program

The function **FN 38: SEND** enables you to write texts and Q parameter values to the log from the NC program and send to a DNC application.

Further information: "FN 16: F-PRINT – Formatted output of text and Q parameter values", Page 201

Data transmission is through a standard TCP/IP computer network.

For more detailed information, consult the Remo Tools SDK manual.

Example

Document values from Q1 and Q23 in the log.

```
FN 38: SEND /"Q parameter Q1: %f Q23: %f" / +Q1 / +Q23
```

9.9 Accessing tables with SQL commands

Introduction

> If you would like to access numerical or alphanumerical content in a table or manipulate the table (e.g., rename columns or rows), then use the SQL commands available to you.
>
> The syntax of the SQL commands available on the control is heavily influenced by the SQL programming language—but does not conform to it completely. In addition, the control does not support the entire scope of the SQL language.
>
> The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when inputting data or reading it out.
>
> The following terms will be used (along with others) in the following:
>
> - "SQL command" refers to the available soft keys
> - "SQL instructions" describe miscellaneous functions that are entered manually as part of the syntax
> - **HANDLE** in the syntax identifies a certain transaction (followed by the parameter for identification)
> - **Result-set** contains the result of the query (known as the result set)

In the NC software, access to tables is gained via an SQL server. This server is controlled with the available SQL commands. The SQL commands can be defined directly in an NC program.

The saver is based on a transaction model. A **transaction** is made up of multiples steps that are executed together, thereby ensuring an orderly and defined processing of the table entries.

> Read- and write-access to individual values of a table can likewise be carried out using the function **FN 26: TABOPEN**, **FN 27: TABWRITE**, and **FN 28: TABREAD**.
> **Further information:** "Freely definable tables", Page 257
>
> HEIDENHAIN recommends using SQL functions instead of **FN 26**, **FN 27**, or **FN 28** with HDR hard disks in order to achieve maximum speeds with table applications and also to reduce the amount of computing power necessary.

> SQL functions can only be tested in the **Program run, single block**, **Program run, full sequence**, and **Positioning with Manual Data Input** modes.

Simplified representation of SQL commands

Example of an SQL transaction:

- Assign Q parameters to table columns for read or write access using **SQL BIND**
- Select data using **SQL EXECUTE** with the instruction **SELECT**
- Read, change, or add data using **SQL FETCH**, **SQL UPDATE**, and **SQL INSERT**
- Confirm or discard interaction using **SQL COMMIT** and **SQL ROLLBACK**
- Approve bindings between table columns and Q parameters using **SQL BIND**

> You must conclude all transactions that have been started—even exclusively read accesses. Concluding the transaction is the only way to ensure that changes and additions are transferred, that locks are removed, and that used resources are released.

Overview of functions

The following table lists all SQL commands available to the user.

Overview of soft keys

Soft key	Command	Page
SQL BIND	**SQL BIND** establishes or removes connections between table columns and Q or QS parameters	216
SQL EXECUTE	**SQL EXECUTE** opens a transaction for selected table columns and table rows or enables the use of other SQL instructions (miscellaneous functions). **Further information:** "Overview of instructions", Page 213	217
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	221
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	227
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	226
SQL UPDATE	**SQL UPDATE** Expands the transaction with a change to the existing row	223
SQL INSERT	**SQL INSERT** creates a new table row	225
SQL SELECT	**SQL SELECT** reads out a single values from a table and does not open any transaction	229

Overview of instructions

The following so-called SQL instructions are used in the SQL command **SQL EXECUTE**.

Further information: "SQL EXECUTE", Page 217

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate a table
COPY TABLE	Copying a table
RENAME TABLE	Rename table
DROP TABLE	Delete the table
INSERT	Inserting table rows
UPDATE	Update the table rows
DELETE	Delete table rows
ALTER TABLE	■ Add table columns using **ADD** ■ Delete table columns using **DROP**
RENAME COLUMN	Rename table columns

> The **result set** describes the result set of a table file. The result set is acquired by a query with **SELECT**.
>
> The **result set** is created when a query is executed in the SQL server, thereby occupying resources there.
>
> This query is like applying a filter to the table, so that only part of the data records become visible. To make this query possible, the table file must be read a this point.
>
> The SQL server assigns a **handle** to the **result set,** which enables you to identify the result set for reading/editing data and completing the transaction. The **handle** is the result of the query, which is visible in the NC program. The value 0 indicates an invalid **handle,** meaning that it was not possible to create a **result set** for that query. If no rows that satisfy the specified condition are found, an empty **result set** is created and assigned a valid **handle.**

Programming SQL commands

> This function is not enabled until the code number **555343** is entered.

You can program SQL commands in the **Programming** operating mode or in **Positioning with mdi**:

SPEC FCT
- Press the **SPEC FCT** key

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

▷
- Shift the soft-key row

SQL
- Press the **SQL** soft key
- Select the SQL command via soft key

> Read and write accesses performed with the help of SQL commands always occur in metric units, regardless of the unit of measure selected for the table or the NC program.
>
> If, for example, a length is saved from one table to a Q parameter, then the value is thereafter always in metric units. If this value is then use in an inch program for the purpose of positioning (**L X+Q1800**), then an incorrect position will be the result.

Example

In the following example, the defined material will be read out from the table (**MILL.TAB**) and saved as text in a QS parameter. The following example shows a possible application and the necessary program steps. Following the examples is recommended for programming of the syntax.

> You can use the **FN 16** function, for example:, in order to reuse QS parameters in your own log files.
>
> **Further information:** "Basics", Page 201

Example for a synonym

0 BEGIN PGM SQL MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\MILL.TAB'"	Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NO==3"	Define search
4 SQL FETCH Q1900 HANDLE QL1	Execute search
5 SQL ROLLBACK Q1900 HANDLE QL1	Complete transaction
6 SQL BIND QS1800	Remove parameter binding
7 SQL Q1 "DROP SYNONYM my_table"	Delete synonym
8 END PGM SQL MM	

Step	Explanation
1 Create synonym	A synonym is assigned to a path (long path names are replaced by short names) ■ The path **TNC:\table\MILL.TAB** must contained in single quotation marks for this. ■ The selected synonym is **my_table**
2 Bind QS parameters	A QS parameter is bound to a table column ■ **QS1800** is freely available in user programs ■ The synonym replaces the entry of the complete path ■ The defined column from the table is called **WMAT**
3 Define search	A search definition contains the entry of the transfer value ■ The **QL1** local parameter (freely selectable) serves to identify the transaction (multiple transactions are possible simultaneously) **QL1**, with the **HANDLE** that designates the transaction, is written here. ■ The synonym defines the table ■ The **WMAT** entry defines the table column of the read operation ■ The entries **NO** and **=3** define the table rows of the read operation ■ Selected table columns and rows define the cells of the read operation
4 Execute search	The read operation is executed ■ **SQL FETCH** is used to copy values from the **result set** into the associated Q parameter or QS parameter. ■ **0** successful read operation ■ **1** faulty read operation ■ The **HANDLE QL1** syntax is the transaction designated by the **QL1** parameter ■ The parameter **Q1900** is a return value for checking whether the data were read.
5 Complete transaction	The transaction is concluded and the used resources are released
6 Remove binding	The binding between table columns and QS parameters is removed (release of necessary resources)
7 Delete synonym	The synonym is deleted again (release of necessary resources)

> The use of synonyms is not obligatory. Instead of a synonym you can also enter the entire path in the SQL commands. Relative path entries are not possible. Following the examples is recommended for programming of the syntax.

In the following NC program the same example is used to explain the entry of absolute paths.

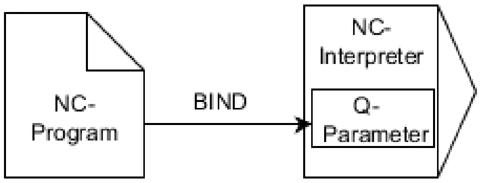
Example for absolute path entries

0 BEGIN PGM SQL_TEST MM	
1 SQL BIND QS 1800 "'TNC:\table\Fraes.TAB'.WMAT"	Bind QS parameters
2 SQL QL1 "SELECT WMAT FROM 'TNC:\table\FRAES.TAB' WHERE NR ==3"	Define search
3 SQL FETCH Q1900 HANDLE QL1	Execute search
4 SQL ROLLBACK Q1900 HANDLE QL1	Complete transaction
5 SQL BIND QS 1800	Remove parameter binding
6 END PGM SQL_TEST MM	

SQL BIND

Example: binding Q parameters to table columns

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
```



Example: remove binding

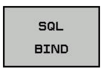
```
91 SQL BIND Q881
92 SQL BIND Q882
93 SQL BIND Q883
94 SQL BIND Q884
```

SQL BIND links a Q parameter to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) 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.

Programming notes:

- You can program any number of bindings. During read and write operations, the only columns taken into consideration are those that are specified using the **SELECT** command. If you specify columns without binding in the **SELECT** command, then the control will interrupt the read or write operation with an error message.
- **SQL BIND...** must be programmed **before** the **FETCH**, **UPDATE**, and **INSERT** commands.



- **Parameter no. for result**: define Q parameter for binding to the table column
- **Database: column name**: define table name and table column (separate with **.**)
  - **Table name**: synonym or path with filename of the table
  - **Column name**: name displayed in the table editor

SQL EXECUTE

SQL EXECUTE is used in connection with various SQL instructions.

Further information: "Overview of instructions", Page 213

SQL EXECUTE with the SQL instruction SELECT

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 (the **INDEX**) is used for the SQL commands **FETCH** and **UPDATE**.

SQL EXECUTE, in combination with the SQL instruction **SELECT**, selects table values and transfers them to the **result set**. In contrast to the SQL command **SQL SELECT**, the combination of **SQL EXECUTE** and the instruction **SELECT** selects multiple columns and rows simultaneously and always opens a transaction.

In the function **SQL ... "SELECT...WHERE..."**, you can enter the search criteria. This lets you restrict the number of rows to be transferred. If you do not use this option, then all of the rows in the table are loaded.

In the function **SQL ... "SELECT...ORDER BY..."**, you can enter the ordering criterion. This entry consists of the column designation and the keyword (**ASC**) for ascending or (**DESC**) for descending order. If you do not use this option, then rows will be stored in a random order.

With the function **SQL ... "SELECT...FOR UPDATE"**, you can lock the selected rows for other applications. Other applications can continue to read these rows but are unable to change them. If you make changes to the table entries, then it is absolutely necessary to use this option.

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

Example: selection of table rows

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example"
```

Example: selection of table rows with the WHERE function

```
. . .
20 SQL Q5 "SELECT Meas_No,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example WHERE Meas_No<20"
```

Example: selection of table rows with the WHERE function and Q parameters

```
. . .
20 SQL Q5 "SELECT Meas_No,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example WHERE Meas_No==:'Q11'"
```

Example: table name defined with path and file name

```
. . .
20 SQL Q5 "SELECT Meas_No,Meas_X,Meas_Y, Meas_Z FROM 'V:
   \table\Tab_Example' WHERE Meas_No<20"
```

SQL EXECUTE

- **Parameter number for result**
  - Return value serves as identification number of a transaction is one was opened
  - The return value serves to check whether the read-process was successful

  The **HANDLE**, which will enable you to access the data at a later date, is stored in the specified parameter. The **HANDLE** is valid until the transaction has been committed or canceled for all rows of the **result set**.
  - **0**: faulty read operation
  - not equal to **0**: return value of the **HANDLE**
- **Database: SQL instruction**: Program an SQL instruction
  - **SELECT** with the table column(s) to be transferred (separate multiple columns with **,**)
  - **FROM** with a table's synonym or path (place the path in single quotation marks)
  - **WHERE** (optional) with column names, condition, and comparison value (Q parameters after **:** in single quotation marks)
  - **ORDER BY** (optional) with column name and type of sorting (**ASC** for ascending order, **DESC** for descending order)
  - **FOR UPDATE** (optional) to lock write access to the selected row for other processes

Conditions for WHERE entries

Condition	Programming
Equals	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
empty	IS NULL
Not empty	IS NOT NULL
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

Syntax examples:

The following examples are listed without context. The NC blocks are limited exclusively to the possibilities of the SQL command **SQL EXECUTE**.

Example

9 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\MILL.TAB'"	Create synonym
9 SQL Q1800 "DROP SYNONYM my_table"	Delete synonym
9 SQL Q1800 "CREATE TABLE my_table (NO,WMAT)"	Create table with the rows NO and WMAT.
9 SQL Q1800 "COPY TABLE my_table TO 'TNC:\table\MILL2.TAB'"	Copy table
9 SQL Q1800 "RENAME TABLE my_table TO 'TNC:\table\MILL3.TAB'"	Rename table
9 SQL Q1800 "DROP TABLE my_table"	Delete the table
9 SQL Q1800 "INSERT INTO my_table VALUES (1,'ENAW',240)"	Insert table row
9 SQL Q1800 "DELETE FROM my_table WHERE NO==3"	Delete table row
9 SQL Q1800 "ALTER TABLE my_table ADD (WMAT2)"	Insert table rows
9 SQL Q1800 "ALTER TABLE my_table DROP (WMAT2)"	Delete table rows
9 SQL Q1800 "RENAME COLUMN my_table (WMAT2) TO (WMAT3)"	Rename table column

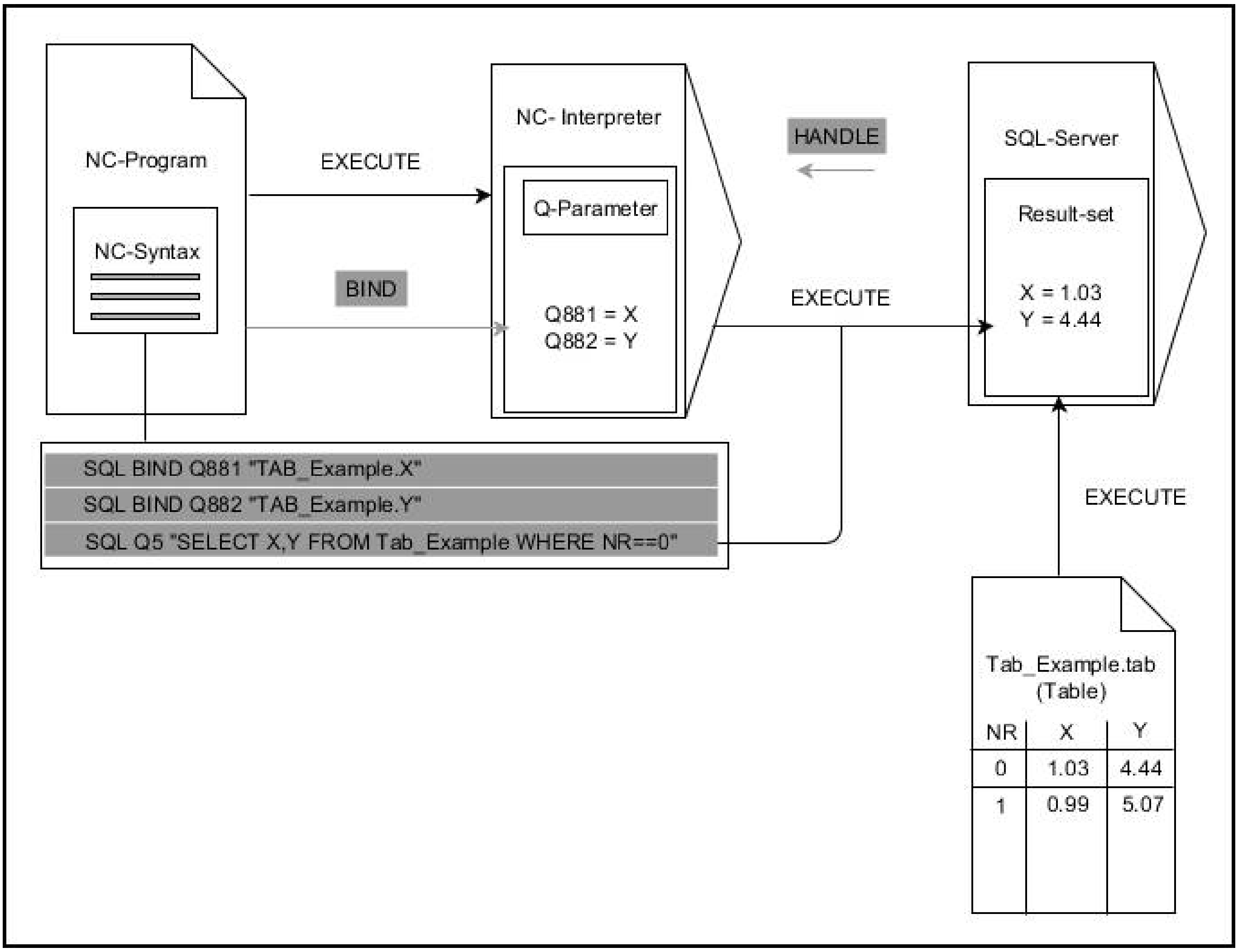
Example:

The following example illustrates the SQL instruction **CREATE TABLE**.

0 BEGIN PGM SQL_TAB_ERSTELLEN_TEST MM	
1 SQL Q10 "CREATE SYNONYM ERSTELLEN FOR 'TNC:\table\ErstellenTab.TAB'"	Create synonym
2 SQL Q10 "CREATE TABLE ERSTELLEN AS SELECT X,Y,Z FROM 'TNC:\prototype_for_erstellen.tab'"	Creates table
3 END PGM SQL_TAB_ERSTELLEN_TEST MM	

A synonym can also be created for a table that has not been created yet.

Example for the **SQL EXECUTE** command:



Gray arrows and associated syntax do not belong directly to the **SQL EXECUTE** command
Black arrows and associated syntax show internal processes of **SQL EXECUTE**

SQL FETCH

Example: transferring row number in the Q parameter

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: programming the row number directly

```
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX5
```

SQL FETCH reads a row from the **result set**. The values of the individual cells are stored in the bound Q parameters. The transaction is defined via the **HANDLE** to be specified; the row is defined via the **INDEX**.

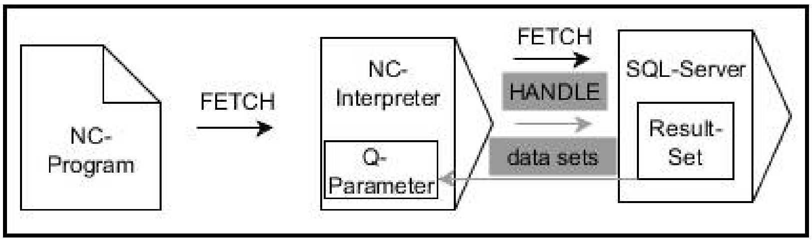
SQL FETCH takes all columns into consideration that were specified with the **SELECT** instruction (SQL command **SQL EXECUTE**).

SQL FETCH

- **Parameter No. for result** (return value for the control):
  - **0** successful read operation
  - **1** faulty read operation
- **Database: SQL access ID**: define Q parameters for the **HANDLE** (for identifying the transaction)
- **Database: index to SQL result**: row number within the **result set**
  - Program the row number directly
  - Program the Q parameter containing the index
  - The row (n=0) is read if nothing is specified

The optional syntax elements **IGNORE UNBOUND** and **UNDEFINE MISSING** are intended for the machine tool builder.

Example for the **SQL FETCH** command:



Gray arrows and associated syntax do not belong directly to the **SQL FETCH** command
Black arrows and associated syntax show internal processes of **SQL FETCH**

SQL UPDATE

Example: transferring row number in the Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MESS_NR"
12 SQL BIND Q882 "TAB_EXAMPLE.MESS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MESS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MESS_Z"
. . .
20 SQL Q5 "SELECT MESS_NR,MESS_X,MESS_Y,MESS_Z FROM
   TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: programming the row number directly

```
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified; the row is defined via the **INDEX**. The existing row in the **result set** is completely overwritten.

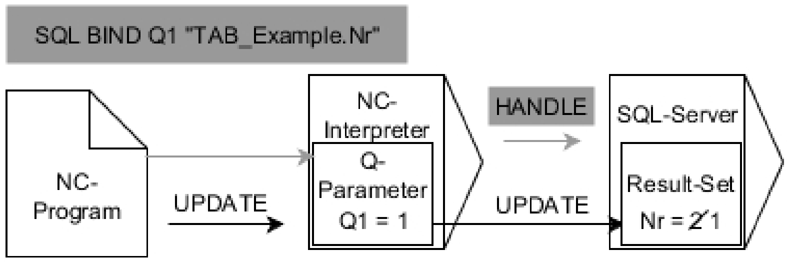
SQL UPDATE takes all columns into consideration that were specified with the **SELECT** instruction (SQL command **SQL EXECUTE**).

SQL UPDATE

- **Parameter No. for result** (return value for the control):
  - **0**: successful change
  - **1**: failed change
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)
- **Database: Index for SQL result**: Row number within the **result set**
  - Program the row number directly
  - Program the Q parameter containing the index
  - The row (n=0) is assigned a value if none is specified

When writing to tables, the control checks the lengths of the string parameters. Error messages are output for entries that would exceed the lengths of the columns to be written to.

Example for the **SQL UPDATE** command:



Gray arrows and associated syntax do not belong directly to the **SQL UPDATE** command
Black arrows and associated syntax show internal processes of **SQL UPDATE**

SQL INSERT

Example: Transferring row number in the Q parameter

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example"
. . .
40 SQL INSERT Q1 HANDLE Q5
```

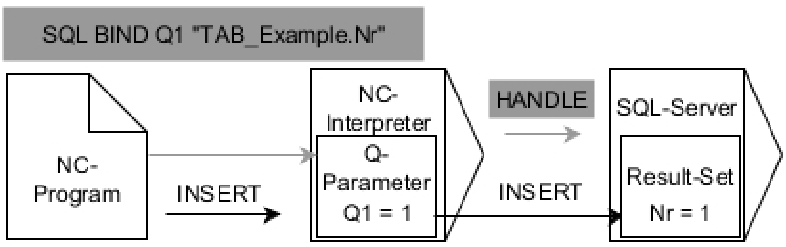
SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified.

SQL INSERT takes all columns into consideration that were specified with the **SELECT** instruction (SQL command **SQL EXECUTE**). Table columns without corresponding **SELECT** instruction (not contained in the query result) are assigned defaults values.

SQL INSERT

- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)

Example for the **SQL INSERT** command:



Gray arrows and associated syntax do not belong directly to the **SQL INSERT**command
Black arrows and associated syntax show internal processes of **SQL INSERT**

When writing to tables, the control checks the lengths of the string parameters. Error messages are output for entries that would exceed the lengths of the columns to be written to.

SQL COMMIT

Example

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
    Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL COMMIT Q1 HANDLE Q5
```

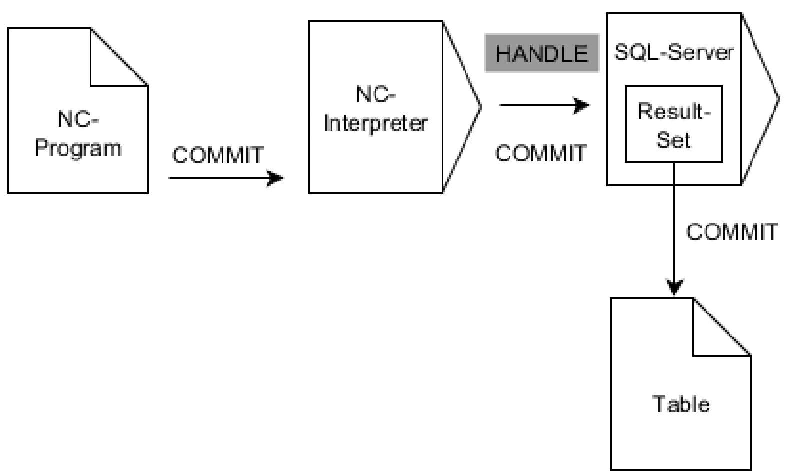
SQL COMMIT simultaneously transfers all of the rows that have been changed and added in a transaction back into the table. The transaction is defined via the **HANDLE** to be specified. A lock that was set with **SELECT...FOR UPDATE** is canceled.

The **HANDLE** (process) assigned with the instruction **SQL SELECT** becomes invalid.

SQL COMMIT

- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)

Example for the **SQL COMMIT** command:



Gray arrows and associated syntax do not belong directly to the **SQL COMMIT**command
Black arrows and associated syntax show internal processes of **SQL COMMIT**

SQL ROLLBACK

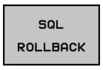
Example

```
11 SQL BIND Q881 "Tab_Example.Meas_No"
12 SQL BIND Q882 "Tab_Example.Meas_X"
13 SQL BIND Q883 "Tab_Example.Meas_Y"
14 SQL BIND Q884 "Tab_Example.Meas_Z"
. . .
20 SQL Q5 "SELECT Meas_no,Meas_X,Meas_Y, Meas_Z FROM
   Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL ROLLBACK Q1 HANDLE Q5
```

SQL ROLLBACK discards all of the changes and additions of a transaction. The transaction is defined via the **HANDLE** to be specified.

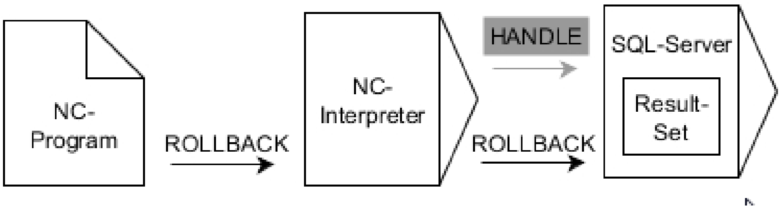
The function of the SQL command **SQL ROLLBACK** depends on the **INDEX**:

- Without **INDEX**:
  - All changes and additions to the transaction are discarded
  - A lock that was set with **SELECT...FOR UPDATE** is canceled.
  - The transaction is concluded (the **HANDLE** loses its validity)
- With **INDEX**:
  - Only the indexed row remains in the **result set** (all other rows are removed)
  - Any changes and additions made in the rows that are not specified are discarded
  - A lock that has been set with **SELECT...FOR UPDATE** remains only for indexed row (all other locks are canceled)
  - The specified (indexed) row becomes the new row 0 of the **result-set**
  - The transaction is **not** concluded (the **HANDLE** keeps its validity)
  - It is necessary to later concluded the transaction using **SQL ROLLBACK** or **SQL COMMIT**



- **Parameter No. for result** (return value for the control):
  - **0** successful transaction
  - **1** successful transaction
- **Database: SQL access ID**: Define Q parameters for the **HANDLE** (for identifying the transaction)
- **Database: Index to SQL result**: Row that remains in the **result set**
  - Program the row number directly
  - Program the Q parameter containing the index

Example for the **SQL ROLLBACK** command:



Gray arrows and associated syntax do not belong directly to the **SQL ROLLBACK** command
Black arrows and associated syntax show internal processes of **SQL ROLLBACK**

SQL SELECT

SQL SELECT reads a single value from a table and saves the result in the defined Q parameter.

> You can select multiple values or columns using the SQL command **SQL EXECUTE** and the **SELECT** instruction.
> **Further information:** "SQL EXECUTE", Page 217

With **SQL SELECT**, there is neither a transaction nor binding between the table columns and Q parameter. Any existing bindings to the specified columns are not taken into consideration; only the read-out value is copied into the parameter specified for the result.

Example: Reading and saving a value

```
20 SQL SELECT Q5 "SELECT Mess_X FROM Tab_Example WHERE
   MESS_NR==3"
```

SQL SELECT

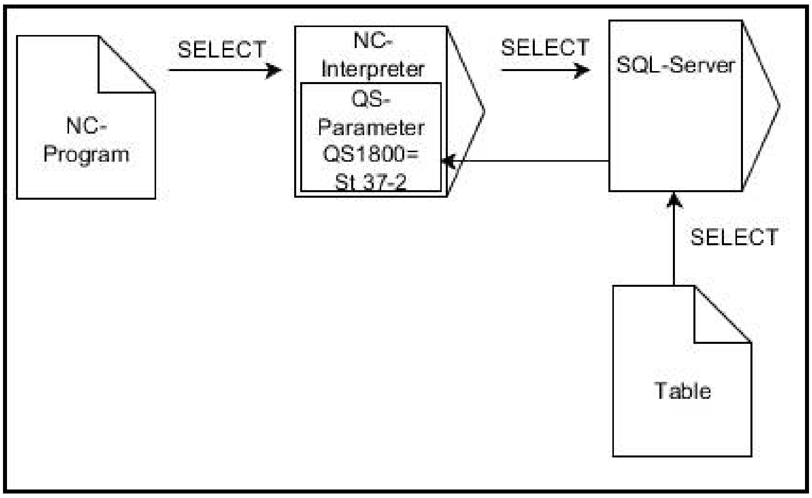
- **Parameter No. for result**: Q parameter for saving the value
- **Database: SQL command text**: Programming SQL instruction
  - **SELECT** with the table column of the value to be transferred
  - **FROM** with a table's synonym or path (place the path in single quotation marks)
  - **WHERE** with column designation, condition and comparison value (Q parameter after **:** in single quotation marks)

The result of the NC program below is identical to the application example shown previously.
Further information: "Example", Page 214

Example

0 BEGIN PGM SQL MM	
1 SQL SELECT QS1800 "SELECT WMAT FROM my_table WHERE NR==3"	Read and save a value
2 END PGM SQL MM	

Example for the **SQL SELECT** command:



Black arrows and associated syntax show internal processes of **SQL SELECT**

9.10 Entering formulas directly

Entering formulas

Using soft keys, you can enter mathematical formulas containing multiple calculation operations directly into the NC program.

Q

- Select Q-parameter functions

FORMULA

- Press the **FORMULA** soft key
- Select **Q**, **QL**, or **QR**

The control displays the following soft keys in several soft-key rows:

Soft key	Linking function
+	**Addition** e. g., **Q10 = Q1 + Q5**
-	**Subtraction** e. g., **Q25 = Q7 - Q108**
*	**Multiplication** e. g., **Q12 = 5 * Q5**
/	**Division** e. g., **Q25 = Q1 / Q2**
(	**Opening parenthesis** e. g., **Q12 = Q1 * (Q2 + Q3)**
)	**Closing parenthesis** e. g., **Q12 = Q1 * (Q2 + Q3)**
SQ	**Square the value** , e.g., **Q15 = SQ 5**
SQRT	**Calculate square root** e.g., **Q22 = SQRT 25**
SIN	**Sine of an angle** e. g., **Q44 = SIN 45**
COS	**Cosine of an angle** e. g., **Q45 = COS 45**
TAN	**Tangent of an angle** e. g., **Q46 = TAN 45**
ASIN	**Arc sine** Inverse function of the sine; determine the angle from the ratio of the opposite side to the hypotenuse e.g., **Q10 = ASIN 0.75**
ACOS	**Arc cosine** Inverse function of the cosine; determine the angle from the ratio of the adjacent side to the hypotenuse e. g., **Q11 = ACOS Q40**

Soft key	Linking function
ATAN	**Arc tangent** Inverse function of the tangent; determine the angle from the ratio of the opposite side to the adjacent side e.g., **Q12 = ATAN Q50**
^	**Powers of values** e. g., **Q15 = 3^3**
PI	**Constant PI (3,14159)** e. g., **Q15 = PI**
LN	**Calculate the natural logarithm of a number** Base 2.7183 e.g., **Q15 = LN Q11**
LOG	**Logarithm of a number, Base 10** e. g., **Q33 = LOG Q22**
EXP	**Exponential function, 2.7183 to the power of n** e. g., **Q1 = EXP Q12**
NEG	**Negate values (multiply by -1)** e.g., **Q2 = NEG Q1**
INT	**Remove digits after the decimal point** Calculate an integer e.g., **Q3 = INT Q42**
ABS	**Absolute value of a number** e. g., **Q4 = ABS Q22**
FRAC	**Remove digits before the decimal point** Calculate a fraction e.g., **Q5 = FRAC Q23**
SGN	**Check algebraic sign of a number** e g., **Q12 = SGN Q50** If return value Q12 = 0, then Q50 = 0 If return value Q12 = 1, then Q50 > 0 If return value Q12 = -1, then Q50 < 0
%	**Calculate modulo value (division remainder)** e. g., **Q12 = 400 % 360** Result: Q12 = 40

The **INT** function does not round off—it simply truncates the decimal places.

Further information: "Example: Rounding a value", Page

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first

Example

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

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

or

Example

```
13 Q2 = SQ 10 - 3^3 = 73
```

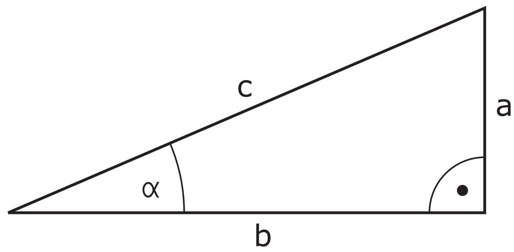
1 Calculation step 10 squared = 100
2 Calculation step 3 to the third power = 27
3 Calculation 100 – 27 = 73

Distributive law

Law of distribution with parentheses calculation
a * (b + c) = a * b + a * c

Example of entry

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



Q
FORMULA

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

Q

- Press the **Q** key on the external alphanumeric keyboard

PARAMETER NUMBER FOR RESULT?

ENT

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

▷
ATAN

- Shift the soft-key row and select the arc tangentfunction

◁
(

- Advance through the soft key menu and press the **OPENING PARENTHESIS** soft key

Q

- Enter **12** (the parameter number)

/

- Select division

Q

- Enter **13** (the parameter number)

)
END

- Close parentheses and conclude formula entry

Example

```
37 Q25 = ATAN (Q12/Q13)
```

9.11 String parameters

String processing functions

You can use the **QS** parameters to create variable character strings. You can output such character strings for example through the **FN 16:F-PRINT** function to create variable logs.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) up to a length of 255 characters to a string parameter. You can also check and process the assigned or imported values using the functions described below. As in Q parameter programming, you can use a total of 2000 QS parameters.

Further information: "Principle and overview of functions", Page 182

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	236
CFGREAD	Read out machine parameter	245
	Chain-linking string parameters	236
TOCHAR	Converting a numerical value to a string parameter	238
SUBSTR	Copy a substring from a string parameter	239
SYSSTR	Read system data	240

Soft key	Formula string functions	Page
TONUMB	Converting a string parameter to a numerical value	241
INSTR	Checking a string parameter	242
STRLEN	Finding the length of a string parameter	243
STRCOMP	Compare alphabetic priority	244

When you use the **STRING FORMULA** function, the result of the arithmetic operation is always a string. When you use the **FORMULA** function, the result of the arithmetic operation is always a numeric value.

Assign string parameters

Before using string variables, you must first assign the variables. Use the **DECLARE STRING** command to do so.

SPEC FCT
- Press the **SPEC FCT** key

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

STRING FUNCTIONS
- Press the **STRING FUNCTIONS** soft key

DECLARE STRING
- Press the **DECLARE STRING** soft key

Example

```
37 DECLARE STRING QS10 = "Workpiece"
```

Chain-linking string parameters

With the concatenation operator (string parameter **||** string parameter) you can make a chain of two or more string parameters.

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

STRING FUNCTIONS

- Press the **STRING FUNCTIONS** soft key

STRING FORMULA

ENT

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the concatenated string. Confirm with the **ENT** key.
- Enter the number of the string parameter in which the **first** substring is saved. Confirm with the **ENT** key
- > The control shows the concatenation symbol **||** an.
- Press the **ENT** key
- Enter the number of the string parameter in which the **second** substring is saved. Confirm with the **ENT** key
- Repeat the process until you have selected all the required substrings. Conclude with the **END** key

Example: QS10 is to include the complete text of QS12, QS13 and QS14

```
37 QS10 =  QS12 || QS13 || QS14
```

Parameter contents:

- **QS12**: **Workpiece**
- **QS13**: **Status:**
- **QS14**: **Scrap**
- **QS10**: **Workpiece Status: Scrap**

Converting a numerical value to a string parameter

With the **TOCHAR** function, the control converts a numerical value into a string parameter. This enables you to chain numerical values with string variables.

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Open the function menu

STRING FUNCTIONS

- Press the String functions soft key

STRING FORMULA

- Press the **STRING FORMULA** soft key

TOCHAR

- Select the function for converting a numerical value to a string parameter
- Enter the number or the desired Q parameter to be converted by the control, and confirm with the **ENT** key
- If desired, enter the number of digits after the decimal point that the control should convert, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

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

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

Copying a substring from a string parameter

The **SUBSTR** function copies a definable range from a string parameter.

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Open the function menu

STRING FUNCTIONS

- Press the String functions soft key

STRING FORMULA

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the character string. Confirm with the **ENT** key.

SUBSTR

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

> The first character of a text string starts internally at the 0-position

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

```
37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```

Reading system data

With the function **SYSSTR** you can read system data and store them in string parameters. You select the system data through a group number (ID) and a number.

Entering IDX and DAT is not required.

Group name, ID no.	Number	Meaning
Program information, 10010	1	Path of the current main program or pallet program
	2	Path of the NC program shown in the block display
	3	Path of the cycle selected with **CYCL DEF 12 PGM CALL**
	10	Path of the NC program selected with **SEL PGM**
Channel data, 10025	1	Channel name
Values programmed in the tool call, 10060	1	Tool name
Current system time, 10321	1 - 16	■ 1: DD.MM.YYYY hh:mm:ss ■ 2 and 16: DD.MM.YYYY hh:mm ■ 3: DD.MM.YY hh:mm ■ 4: YYYY-MM-DD hh:mm:ss ■ 5 and 6: YYYY-MM-DD hh:mm ■ 7: YY-MM-DD hh:mm ■ 8 and 9: DD.MM.YYYY ■ 10: DD.MM.YY ■ 11: YYYY-MM-DD ■ 12: YY-MM-DD ■ 13 and 14: hh:mm:ss ■ 15: hh:mm
Touch-probe data, 10350	50	Probe type of the active touch probe TS
	70	Probe type of the active touch probe TT
	73	Key name of the active touch probe TT from MP **activeTT**
	2	Path of the selected pallet table
NC software version, 10630	10	Version identifier of the NC software version
Tool data, 10950	1	Tool name
	2	DOC entry of the tool
	4	Tool-carrier kinematics

Converting a string parameter to a numerical value

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

> The QS parameter to be converted must contain only one numerical value. Otherwise, the Control will output an error message..

Q
- Select Q-parameter functions

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

◁
- Shift the soft-key row

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

Example: Convert string parameter QS11 to a numerical parameter Q82

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

Testing a string parameter

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

Q
- Select Q-parameter functions

FORMULA
- Press the **FORMULA** soft key
- Enter the number of the Q parameter for the result and confirm with the **ENT** key
- > The control saves the place at which the text to be searched for begins. It is saved in the parameter.

◁
- Shift the soft-key row

INSTR
- Select the function for checking a string parameter
- Enter the number of the QS parameter in which the text to be searched for is saved. Confirm with the **ENT** key
- Enter the number of the QS parameter to be searched for by the control, and confirm with the **ENT** key
- Enter the number of the place at which the control is to start search the substring, and confirm with the **ENT** key.
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

> The first character of a text string starts internally at the 0-position
>
> If the control cannot find the required substring, it will save the total length of the string to be searched (counting starts at 1) in the result parameter.
>
> If the substring to be searched for appears multiple times, then the control returns the first place at which it finds the substring.

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

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

Finding the length of a string parameter

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

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the ascertained string length. Confirm with the **ENT** key.

◁

- Shift the soft-key row

STRLEN

- Select the function for finding the text length of a string parameter
- Enter the number of the QS parameter from which the control is to ascertain the length, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Find the length of QS15

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

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

Comparing alphabetic priority

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

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the result of comparison, and confirm with the **ENT** key.

◁

- Shift the soft-key row

STRCOMP

- Select the function for comparing string parameters
- Enter the number of the first QS parameter that the control is to compare, and confirm with the **ENT** key
- Enter the number of the second QS parameter that the control is to compare, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

The control returns the following results:

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

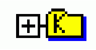
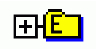
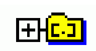
Example: QS12 and QS14 are compared for alphabetic priority

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

Reading out machine parameters

With the **CFGREAD** function, you can read out machine parameters of the control as numerical values or as strings. The read-out values are always output in metric units of measure.

In order to read out a machine parameter, you must use the control's configuration editor to determine the parameter name, parameter object, and, if they have been assigned, the group name and index:

Icon	Type	Meaning	Example
	**Key**	Group name of the machine parameter (if available)	CH_NC
	**Entity**	Parameter object (name begins with **Cfg**...)	**CfgGeoCycle**
	Attribute	Name of the machine parameter	**displaySpindleErr**
	**Index**	List index of a machine parameter (if available)	[0]

> If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts.
>
> **Further information:** User's Manual for Setup, Testing and Running NC Programs

Each time you want to interrogate a machine parameter with the **CFGREAD** function, you must first define a QS parameter with attribute, entity and key.

The following parameters are read in the CFGREAD function's dialog:

- **KEY_QS**: Group name (key) of the machine parameter
- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

Reading a string of a machine parameter

In order to store the content of a machine parameter as a string in a QS parameter:

Q

- Press the **Q** key.

STRING FORMULA

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the machine parameter
- Press the **ENT** key
- Select the **CFGREAD** function
- Enter the numbers of the string parameters for key, entity, and attribute
- Press the **ENT** key
- Enter the number for the index, or skip the dialog with N**NO ENT**, whichever applies
- Close the parenthesized expression with the **ENT** key
- Press the **END** key to conclude entry

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

Parameter settings in the configuration editor
DisplaySettings
CfgDisplayData
axisDisplayOrder
[0] to [3]

Example

14 QS11 = ""	Assign string parameter for key
15 QS12 = "CfgDisplaydata"	Assign string parameter for entity
16 QS13 = "axisDisplay"	Assign string parameter for parameter name
17 QS1 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13 IDX3)	Read out machine parameter

Reading a numerical value of a machine parameter

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

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the machine parameter
- Press the **ENT** key
- Select the **CFGREAD** function
- Enter the numbers of the string parameters for key, entity, and attribute
- Press the **ENT** key
- Enter the number for the index, or skip the dialog with N**NO ENT**, whichever applies
- Close the parenthesized expression with the **ENT** key
- Press the **END** key to conclude entry

Example: Read overlap factor as Q parameter

Parameter settings in the configuration editor

ChannelSettings
CH_NC
CfgGeoCycle
pocketOverlap

Example

14 QS11 = "CH_NC"	Assign string parameter for key
15 QS12 = "CfgGeoCycle"	Assign string parameter for entity
16 QS13 = "pocketOverlap"	Assign string parameter for parameter name
17 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	Read out machine parameter

9.12 Preassigned Q parameters

The Q parameters Q100 to Q199 are assigned values by the control. The following types of information are assigned to the Q parameters:

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

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

NOTICE

Danger of collision!

HEIDENHAIN cycles, manufacturer cycles and third-party functions use Q parameters. You can also program Q parameters within NC programs. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine tool builder, and suppliers.
- Check the machining sequence using a graphic simulation

You must not use preassigned Q parameters (QS parameters) between **Q100** and **Q199** (**QS100** and **QS199**) as calculation parameters in the NC programs.

Values from the PLC: Q100 to Q107

The control assigns values from the PLC to parameters Q100 to Q107 in an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or **TOOL DEF** block)
- Delta value DR from the tool table
- Delta value DR from the **TOOL CALL** block

The control remembers the current tool radius even if the power is interrupted.

Tool axis: Q109

The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = –1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

Spindle status: Q110

The value of the parameter Q110 depends on the M function last programmed for the spindle.

M function	Parameter value
No spindle status defined	Q110 = –1
M3: Spindle ON, clockwise	Q110 = 0
M4: Spindle ON, counterclockwise	Q110 = 1
M5 after M3	Q110 = 2
M5 after M4	Q110 = 3

Coolant on/off: Q111

M function	Parameter value
M8: Coolant ON	Q111 = 1
M9: Coolant OFF	Q111 = 0

Overlap factor: Q112

The control assigns Q112 to the overlap factor for pocket milling.

Unit of measurement for dimensions in the NC program: Q113

During nesting the **PGM CALL**, the value of the parameter Q113 depends on the dimensional data of the NC program from which the other NC programs are called.

Dimensional data of the main program	Parameter value
Metric system (mm)	Q113 = 0
Imperial system (inch)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

> The Control remembers the current tool length even if the power is interrupted.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe. The coordinates refer to the 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, for example, the TT 160

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

10

Special Functions

10.1 Overview of special functions

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

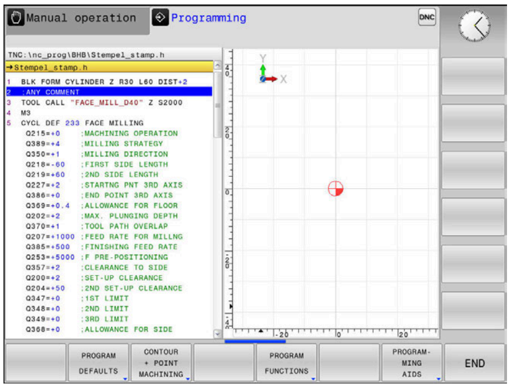
Function	Description
Working with text files	Page 270
Working with freely definable tables	Page 257

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

Main menu for SPEC FCT special functions

SPEC FCT

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



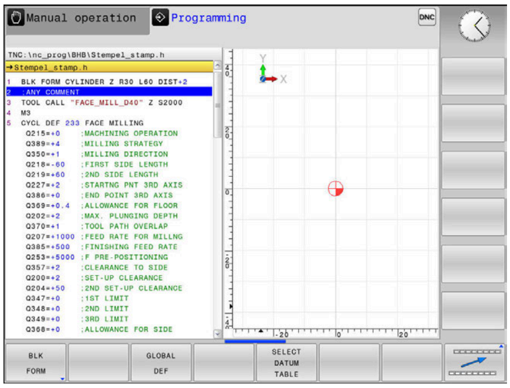
Soft key	Function	Description
PROGRAM DEFAULTS	Define program defaults	Page 253
CONTOUR + POINT MACHINING	Functions for contour and point machining	Page 253
PROGRAM FUNCTIONS	Define different conversational functions	Page 254
PROGRAM-MING AIDS	Programming aids	Page 117

After pressing the **SPEC FCT** key, you can open the **smartSelect** selection window with the **GOTO** key. The control displays a structure overview with all available functions. You can rapidly navigate with the cursor or mouse and select functions in the tree diagram. The control displays online help for the selected function in the window on the right.

Program defaults menu

PROGRAM DEFAULTS

▶ Press the Program Defaults soft key

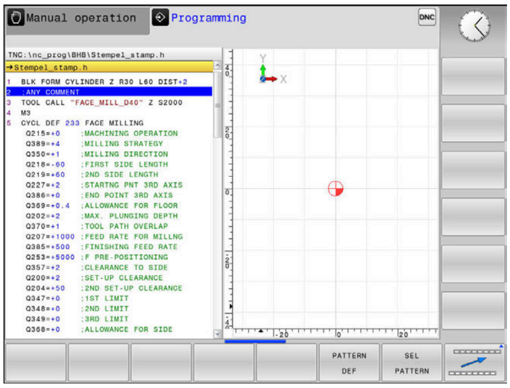


Soft key	Function	Description
BLK FORM	Define workpiece blank	Page 69
DATUM TABLE	Select datum table	Page 392
GLOBAL DEF	Define global cycle parameters	Page 292

Functions for contour and point machining menu

CONTOUR + POINT MACHINING

▶ Press the soft key for functions for contour and point machining

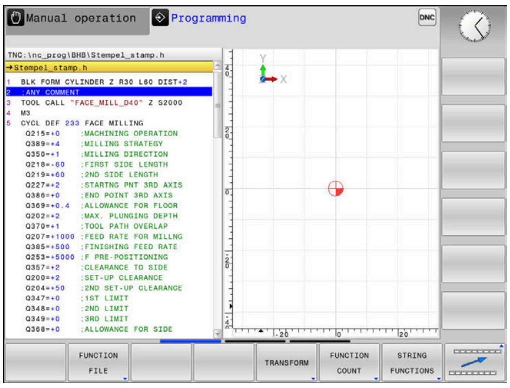


Soft key	Function	Description
PATTERN DEF	Define regular machining pattern	Page 296
SEL PATTERN	Select the point file with machining positions	Page 309

Menu for defining different conversional functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key



Soft key	Function	Description
FUNCTION FILE	Define file functions	Page 266
TRANSFORM / CORRDATA	Define coordinate transforma-tions	Page 267
FUNCTION COUNT	Define the counter	Page 255
STRING FUNCTIONS	Define string functions	Page 235
FUNCTION SPINDLE	Define pulsing spindle speed	Page 262
FUNCTION FEED	Define recurring dwell time	Page 264
FUNCTION DWELL	Define dwell time in seconds or revolutions	Page 279
INSERT COMMENT	Add comments	Page 121

10.2 Defining a counter

Application

Refer to your machine manual.
Your machine manufacturer enables this function.

The **FUNCTION COUNT** function allows you to control a simple counter from within the NC program. For example, this function allows you to count the number of manufactured workpieces.

Proceed as follows for the definition:

SPEC FCT

- Show the soft key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION COUNT

- Press the **FUNCTION COUNT** soft key

NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.

- Please check prior to machining whether a counter is active.
- If necessary, note down the counter value and enter it again via the MOD menu after execution.

Effect in the Test Run operating mode

You can simulate the counter in the **Test Run** operating mode. Only the count you have defined directly in the NC program is effective. The count in the MOD menu remains unaffected.

Effect in the Program Run Single Block and Program Run Full Sequence operating modes

The count from the MOD menu is only effective in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.

The count is retained even after a restart of the control.

Define FUNCTION COUNT

The **FUNCTION COUNT** function provides the following possibilities:

Soft key	Meaning
FUNCTION COUNT INC	Increase count by 1
FUNCTION COUNT RESET	Reset counter
FUNCTION COUNT TARGET	Set the nominal count (target value) to the desired value Input value: 0–9999
FUNCTION COUNT SET	Set the counter to the desired value Input value: 0–9999
FUNCTION COUNT ADD	Increment the counter by the desired value Input value: 0–9999
FUNCTION COUNT REPEAT	Repeat the NC program starting from this label if more parts are to be machined.

Example

5 FUNCTION COUNT RESET	Reset the counter value
6 FUNCTION COUNT TARGET10	Enter the target number of parts to be machined
7 LBL 11	Enter the jump label
8 ...	Machining
51 FUNCTION COUNT INC	Increment the counter value
52 FUNCTION COUNT REPEAT LBL 11	Repeat the machining operations if more parts are to be machined.
53 M30	
54 END PGM	

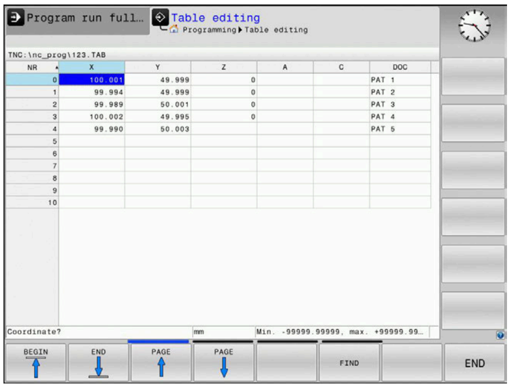
10.3 Freely definable tables

Fundamentals

In freely definable tables you can save and read any information from the NC program. The Q parameter functions **FN 26** to **FN 28** are provided for this purpose.

You can change the format of freely definable tables, i.e. the columns and their properties, by using the structure editor. They enable you to make tables that are exactly tailored to your application.

You can also toggle between a table view (standard setting) and form view.



> The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when inputting data or reading it out.

Creating a freely definable table

Proceed as follows:

PGM MGT

- Press the **PGM MGT** key
- Enter any desired file name with the extension .TAB

ENT

- 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**

ENT

- Confirm with the **ENT** key
- > The control opens a new table in the predefined format.
- To adapt the table to your requirements you have to edit the table format
  Further information: "Editing the table format", Page 258

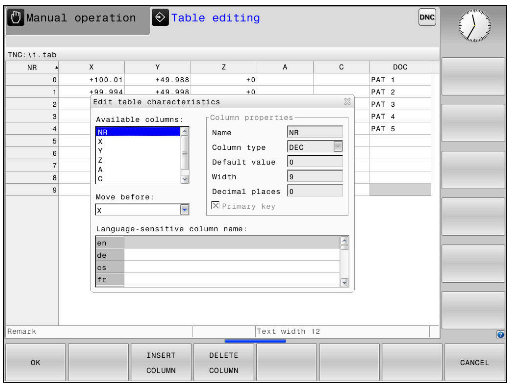
> Refer to your machine manual.
> Machine tool builders may define their own table templates and save them in the control. When you create a new table, the control opens a pop-up window listing all available table templates.

> You can also save your own table templates in the TNC. To do so, create a new table, change the table format and save the table in the **TNC:\system\proto** directory. If you then create new table, the control offers your template in the selection window for table templates.

Editing the table format

Proceed as follows:

- ▶ Press the **EDIT FORMAT** soft key
- \> The control opens a pop-up window displaying the table structure.
- ▶ Adapt the format



The control provides the following options:

Structure command	Meaning
Available columns:	List of all columns contained in the table
Move before:	The entry highlighted in **Available columns** is moved in front of this column
Name	Column name: Is displayed in the header
Column type	**TEXT**: Text entry **SIGN**: + or - sign **BIN**: Binary number **DEC**: Decimal, positive, whole number (cardinal number) **HEX**: Hexadecimal number **INT**: Whole number **LENGTH**: Length (is converted in inch programs) **FEED**: Feed rate (mm/min or 0.1 inch/min) **IFEED**: Feed rate (mm/min or inch/min) **FLOAT**: Floating-point number **BOOL**: Logical value **INDEX**: Index **TSTAMP**: Fixed format for date and time **UPTEXT**: Text entry in upper case **PATHNAME**: Path name
Default value	Default value for the fields in this column
Width	Width of the column (number of charac-ters)
Primary key	First table column
Language-sensitive column name	Language-sensitive dialogs

> ℹ Columns with a column type that permits letters, such as **TEXT**, can only be output or written to via QS parameters, even if the content of the cell is a number.

You can use a connected mouse or the navigation keys to move through the form.

Proceed as follows:

- Press the navigation keys to jump to the input fields

GOTO

- Press the **GOTO** key in order to open expandable menus
- Use the arrow keys to navigate within an input field

> In a table that already contains lines you can not change the table properties **Name** and **Column type**. Once you have deleted all lines, you can change these properties. If required, create a backup copy of the table beforehand.
>
> With the **CE** and **ENT** key combination, you can reset invalid values in fields with the **TSTAMP** column type.

Close the structure editor

Proceed as follows:

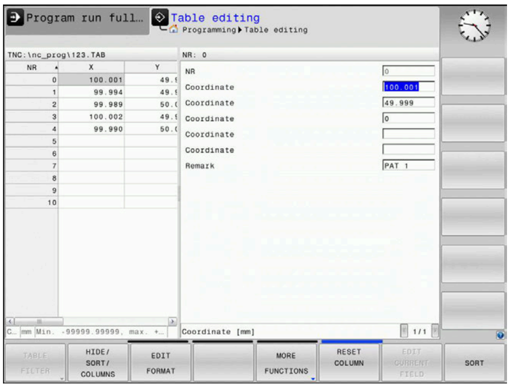
OK

- Press the **OK** soft key
- > The control closes the editing form and applies the changes.

CANCEL

- Alternative: Press the **CANCEL** soft key
- > The control discards all entered changes.

Switching between table and form view



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

Switch the view as follows:

- Press the **Screen layout** key

FORM

- Press the soft key with the desired view

In the left half of the form view, the control lists the line numbers with the contents of the first column.

You can change the data as follows in the form view:

ENT

- Press the **ENT** key in order to switch to the next input field on the right-hand side

Selecting another row to be edited:

- Press the **Next tab** key
- > The cursor jumps to the left window.
- Use the arrow keys to select the desired row
- Press the **Next tab** key to switch back to the input window

FN 26: TABOPEN – Open a freely definable table

With the function **FN 26: TABOPEN** you open a freely definable table to be written to with **FN 27** or to be read from with **FN 28**.

> Only one table can be opened in an NC program at any one time. A new NC block with **FN 26: TABOPEN** automatically closes the last opened table.
>
> The table to be opened must have the extension **.TAB**.

Example: Open the table TAB1.TAB, which is saved in the directory TNC:\DIR1.

```
56 FN 26: TABOPEN TNC:\DIR1\TAB1.TAB
```

FN 27: TABWRITE – Write to a freely definable table

With the **FN 27: TABWRITE** function you write to the table that you previously opened with **FN 26: TABOPEN**.

You can define multiple column names in a **TABWRITE** block. The column names must be written between quotation marks and separated by a comma. You define in Q parameters the value that the control is to write to the respective column.

> The **FN 27: TABWRITE** function by default writes values to the currently open table, even in the **Test Run** operating mode. The **FN 18 ID992 NR16** function allows you to retrieve the operating mode in which the NC program is running. If the function **FN 27** may only be run in the operating modes **Program run, single block** and **Program run, full sequence**, then you can use a jump instruction to skip the corresponding program section.
>
> **Further information:** "If-then decisions with Q parameters", Page 192
>
> If you write to more than one column in an NC block, you must save the values under successive Q parameter numbers.
>
> The control displays an error message if you try to write to a table cell that is locked or does not exist.

Use QS parameters if you want to write to a text field (such as column type **UPTEXT**). Use Q, QL, or QR parameters to write to numerical fields.

Example

You wish to write to the columns “Radius”, “Depth”, and “D” in line 5 of the presently opened table. The values to be written in the table are saved in the Q parameters **Q5**, **Q6**, and **Q7**.

```
53 Q5 = 3.75
54 Q6 = -5
55 Q7 = 7.5
56 FN 27:  TABWRITE 5/"RADIUS,DEPTH,D" = Q5
```

FN 28: TABREAD – Read from a freely definable table

With the **FN 28: TABREAD** function you read from the table previously opened with **FN 26: TABOPEN**.

You can define, i.e. read, multiple column names in a **TABREAD** block. The column names must be written between quotation marks and separated by a comma. In the **FN 28** block you can define the Q parameter number in which the control is to write the value that is first read.

> If you wish to read from more than one column in an NC block, the control will save the values under successive Q parameters of the same time, such as **QL1**, **QL2**, and **QL3**.

Use QS parameters if you want to read a text field. Use Q, QL, or QR parameters to read from numerical fields.

Example

You wish to read the values of the columns **X**, **Y**, 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**).

From the same row, save the column **DOC** in **QS1**.

```
56 FN 28: TABREAD Q10 = 6/“X,Y,D“
57 FN 28: TABREAD QS1 = 6/“DOC“
```

Adapting the table format

NOTICE

Caution: Data may be lost!

The **ADAPT NC PGM / TABLE** function changes the format of all tables permanently. Existing data is not automatically backed up by the control before running the format change process, This permanently changes the files so that they may no longer be usable.

- Only use the function in consultation with the machine tool builder.

Soft key	Function
ADAPT NC PGM / TABLE	Adapt format of tables present after changing the control software version

> The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when inputting data or reading it out.

10.4 Pulsing spindle speed FUNCTION S-PULSE

Programming a pulsing spindle speed

Application

> Refer to your machine manual.
> Read and note the functional description of the machine tool builder.
> Follow the safety precautions.

Using the **S-PULSE FUNCTION** you can program a pulsing spindle speed, when operating at a constant spindle speed.

You can define the duration of a vibration (period length) using the P-TIME input value or a speed change in percent using the SCALE input value. The spindle speed changes in a sinusoidal form around the target value.

Procedure

Example

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

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

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

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

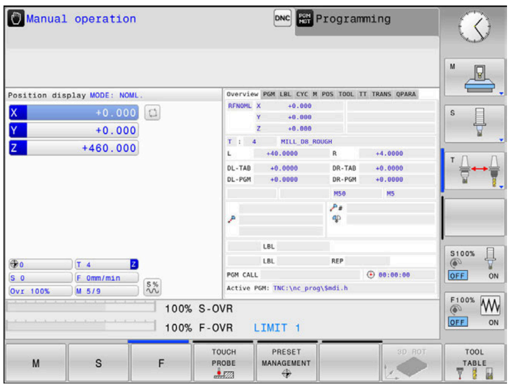
SPINDLE-PULSE
- Press the **SPINDLE-PULSE** soft key
- Define period length P-TIME
- Define speed change SCALE

> The control never exceeds a programmed speed limit. The spindle speed is maintained until the sinusoidal curve of the **S-PULSE FUNCTION** falls below the maximum speed once more.

Symbols

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

Icon	Function
	Pulsing spindle speed active



Resetting the pulsing spindle speed

Example

18 FUNCTION S-PULSE RESET

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

Proceed as follows for the definition:

SPEC FCT	▶ Show the soft-key row with special functions
PROGRAM FUNCTIONS	▶ Press the **PROGRAM FUNCTIONS** soft key
FUNCTION SPINDLE	▶ Press the **FUNCTION SPINDLE** soft key
RESET SPINDLE-PULSE	▶ Press the **RESET SPINDLE-PULSE** soft key.

10.5 Dwell time FUNCTION FEED

Programming dwell time

Application

	Refer to your machine manual. Read and note the functional description of the machine tool builder. Follow the safety precautions.

The **FUNCTION FEED DWELL** function can be used to program a recurring dwell time in seconds, e.g. to force chip breaking . Program **FUNCTION FEED DWELL** immediately prior to the machining you wish to run with chip breaking.

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motion.

NOTICE

Caution: Danger to the tool and workpiece!

When the **FUNCTION FEED DWELL** function is active, the control will repeatedly interrupt the feed movement. While the feed movement is interrupted, the tool remains at its current position while the spindle continues to turn. Due to this behavior, workpieces need to be scrapped if threads are cut. In addition, there is a danger of tool breakage during execution!

- Deactivate the **FUNCTION FEED DWELL** function before cutting threads

Procedure

Example

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

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

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

FUNCTION FEED
- Press the **FUNCTION FEED** soft key

FEED DWELL
- Press the **FEED DWELL** soft key
- Define the interval duration for dwelling D-TIME
- Define the interval duration for cutting F-TIME

Resetting dwell time

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

Example

18 FUNCTION FEED DWELL RESET

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

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

FUNCTION FEED
- Press the **FUNCTION FEED** soft key

RESET FEED DWELL
- Press the **RESET FEED DWELL** soft key

> You can also reset the dwell time by entering D-TIME 0.
> The control automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

10.6 File functions

Application

The **FILE FUNCTION** functions are used to perform file operations such as copying, moving, and deleting files from within the NC program.

> You must not use **FILE** functions on NC programs or files, to which you have previously made reference with functions such as **CALL PGM** or **CYCL DEF 12 PGM CALL**.

Defining file functions

SPEC FCT
- Press the special functions key

PROGRAM FUNCTIONS
- Select the program functions

FUNCTION FILE
- Select file operations
- > The control displays the available functions.

Soft key	Function	Meaning
FILE COPY	**FILE COPY**	Copy file: Enter the name and path of the file to be copied, as well as the target path
FILE MOVE	**FILE MOVE**	Move file: Enter the name and path of the file to be moved, as well as the target path
FILE DELETE	**FILE DELETE**	Delete file: Enter the path and name of the file to be deleted

If you try to copy a file that does not exist, the control generates an error message.

FILE DELETE does not generate an error message if you try to delete a non-existing file.

10.7 Defining coordinate transformations

Overview

As an alternative to the coordinate transformation Cycle 7, **DATUM SHIFT**, you can also use the **TRANS DATUM** conversational function. Just as in Cycle 7, you can use **TRANS DATUM** to directly program shift values or activate a line from a selectable datum table. In addition, there is also the **TRANS DATUM RESET** function that can be used to easily reset a datum shift.

TRANS DATUM AXIS

Example

```
13 TRANS DATUM AXIS X+10 Y+25 Z+42
```

You can define a datum shift by entering values in the respective axis with the **TRANS DATUM AXIS** function. You can define up to nine coordinates in one NC block, and incremental entries are possible. Proceed as follows for the definition:

SPEC FCT	▶ Show the soft-key row with special functions
PROGRAM FUNCTIONS	▶ Press the **PROGRAM FUNCTIONS** soft key
TRANSFORM / CORRDATA	▶ Select transformations
TRANS DATUM	▶ Select the **TRANS DATUM** datum shift
VALUES X Y Z	▶ Select the value input soft key ▶ Enter the datum shift in the affected axes, confirming with the **ENT** key each time

> Values entered as absolute numbers refer to the workpiece preset, which is specified either by presetting or by selecting a preset from the preset table.
>
> Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted).

TRANS DATUM TABLE

Example

```
13 TRANS DATUM TABLE TABLINE25
```

You can define a datum shift by selecting a datum number from a datum table with the **TRANS DATUM TABLE** function. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

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

TRANSFORM / CORRDATA
- Select transformations

TRANS DATUM
- Select the **TRANS DATUM** datum shift

TABLE .D
- Select the **TRANS DATUM TABLE** datum shift
- Enter the line number to be activated by the control, confirm with the **ENT** key
- If desired, enter the name of the datum table from which you want to activate the datum number, and confirm with the **ENT** key. If you do not want to define a datum table, confirm with the **NO ENT** key

> If you have not defined a datum table in the **TRANS DATUM TABLE** block, then the control uses the datum table previously selected with **SEL TABLE** or the datum table activated in the **Program run, single block** or **Program run, full sequence** operating mode (status **M**).

TRANS DATUM RESET

Example

```
13 TRANS DATUM  RESET
```

Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant. Proceed as follows for the definition:

SPEC FCT
- Show the soft-key row with special functions

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

TRANSFORM / CORRDATA
- Select transformations

TRANS DATUM
- Select the **TRANS DATUM** datum shift

RESET DATUM SHIFT
- Press the **RESET DATUM SHIFT** soft key

10.8 Creating text files

Application

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

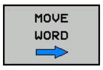
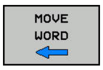
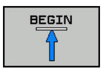
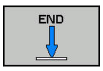
- Recording test results
- Documenting working procedures
- Creating formula collections

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

Opening and exiting a text file

- Operating mode: Press the **Programming** key
- To call the file manager, press the **PGM MGT** key.
- Display type .A files: Press the **SELECT TYPE** soft key and **SHOW ALL** soft key one after the other
- Select a file and open it with the **SELECT** soft key or **ENT** key, or create a new file by entering the new file name and confirming your entry with the **ENT** key

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

Soft key	Cursor movements
	Move cursor one word to the right
	Move cursor one word to the left
	Cursor at beginning of file
	Cursor at end of file

Editing texts

Above the first line of the text editor, there is an information field showing the file name, location and line information:

File:	Name of the text file
Line:	Line in which the cursor is presently located
Column:	Column in which the cursor is presently located

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

You can insert a line break with the **RETURN** or **ENT** key.

Deleting and re-inserting characters, words and lines

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

- Move the cursor to the word or line that you wish to erase and insert at a different place in the text
- Press the **DELETE WORD** or **DELETE LINE** soft key: The text is deleted and stored temporarily.
- Move the cursor to the location where you wish insert the text, and press the **INSERT LINE / WORD** soft key.

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

Editing text blocks

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

- To select a text block: Move the cursor to the first character of the text you wish to select.

SELECT BLOCK

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

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

Soft key	Function
CUT OUT BLOCK	Delete the selected block and store temporarily
COPY BLOCK	Store the selected block temporarily without erasing (copy)

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

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

INSERT BLOCK

- Press the **INSERT BLOCK** soft key—the text block is inserted.

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

Transferring the selected block to a different file

- Select the text block as described previously

APPEND TO FILE

- Press the **APPEND TO FILE** soft key.
- > The control displays the **file name** dialog message.
- Enter the path and the name of the destination file.
- > The control appends the selected text block to the specified file.

Inserting another file at the cursor position

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

READ FILE

- Press the **READ FILE** soft key.
- > The control displays the **File name =** dialog message.
- Enter the path and name of the file you want to insert

Finding text sections

With the text editor, you can search for words or character strings in a text. The control provides the following two options.

Finding the current text

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

- Move the cursor to the desired word.
- To select the search function, press the **FIND** soft key.
- Press the **FIND CURRENT WORD** soft key.
- To find a word: press the **FIND** soft key.
- Exit the search function: Press the **END** soft key

Finding any text

- To select the search function, press the **FIND** soft key. The control 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

10.9 Tool carrier management

Fundamentals

You can create and manage tool carriers using the tool carrier management. The control factors the tool carriers into the calculations.

On machines with 3 axes, tool carriers for right-angled angle heads help processing on tool axes **X** and **Y**, as the control takes the dimensions of the angle heads into consideration.

You must carry out the following steps so that the control can factors the tool carriers into the calculations:

- Save tool carrier templates
- Assign input parameters to tool carriers
- Allocate parameterized tool carriers

Saving tool carrier templates

Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. So that you don't have to design all your tool carriers yourself, HEIDENHAIN supplies a range of ready-made tool carrier templates. Tool carrier templates are 3-D models with fixed geometries but changeable dimensions.

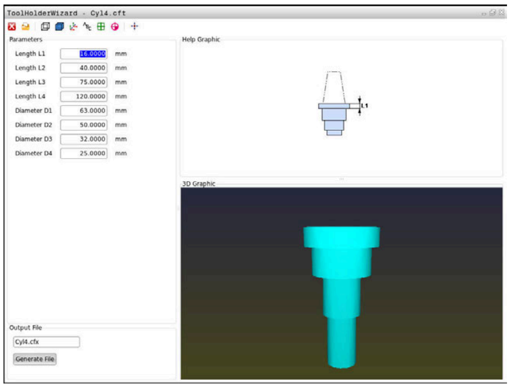
The tool carrier templates must be saved in **TNC:\system \Toolkinematics** and have the extension **.cft**.

> If the tool carrier templates are not available in your control, please download the data you require from:
>
> **http://www.klartext-portal.com/nc-solutions/en**

> If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

> The tool carrier templates may consist of several sub-files. If the sub-files are incomplete, the control will display an error message.
>
> **Do not use incomplete tool carrier templates!**

Assigning input parameters to tool carriers

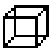
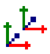
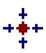


Before the control can factor the tool carrier into the calculations, you must give the tool carrier template the actual dimensions. These parameters are entered in the additional **ToolHolderWizard** tool.

Save the parameterized tool carriers with the extension **.cfx** under **TNC:\system\Toolkinematics**.

The additional **ToolHolderWizard** tool is mainly operated with a mouse. Using the mouse, you can also set the desired screen layout by drawing a line between the areas **Parameter**, **Help graphics** and **3-D graphics** by holding down the left mouse button.

The following icons are available in the additional **ToolHolderWizard** tool:

Icon	Function
	Close tool
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Display or hide transformation vectors
	Show or hide names of collision objects
	Display or hide test points
	Show or hide measurement points
	Return to starting view of the 3-D model

> 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.

Parameterize the tool carrier template in the Manual operation mode

Proceed as follows to parameterize tool carrier templates and save these parameters:

- Press the **Manual operation** key
- Press the **TOOL TABLE** soft key
- Press the **EDIT** soft key
- Move the cursor to the **KINEMATIC** column
- Press the **SELECT** soft key
- Press the **TOOL HOLDER WIZARD** soft key
- > The control opens the additional **ToolHolderWizard** tool in a pop-up window.
- Press the **OPEN FILE** icon
- > The control opens a pop-up window.
- Select the desired tool carrier template using the preview screen
- Press the **OK** button
- > The control opens the selected tool carrier template.
- > The cursor goes to the first parameterizable value.
- Adjust values
- Enter the name for the parameterized tool holder in the **Output file** area
- Press the **GENERATE FILE** button
- If required, reply to the message on the control
- Press the **CLOSE** icon
- > The control closes the additional tool

Parameterize the tool carrier template in the Programming operating mode

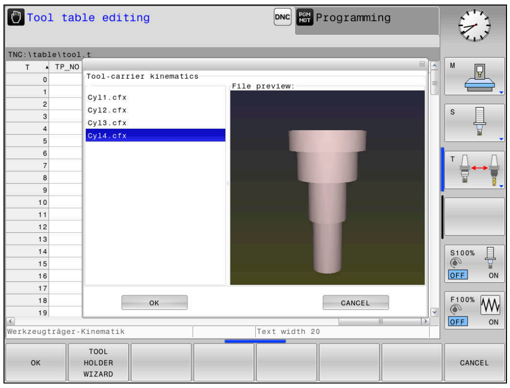
Proceed as follows to parameterize tool carrier templates and save these parameters:

- Press the **Programming** key
- Press the **PGM MGT** key
- Select the path **TNC:\system\Toolkinematics**
- Select the tool carrier template
- > The control opens the additional **ToolHolderWizard** tool with the selected tool carrier template.
- > The cursor goes to the first parameterizable value.
- Adjust values
- Enter the name for the parameterized tool holder in the **Output file** area
- Press the **GENERATE FILE** button
- If required, reply to the message on the control
- Press the **CLOSE** icon
- > The control closes the additional tool

Allocating parameterized tool carriers

To allow the control to factor a parameterized tool carrier into calculations, you must allocate the tool carrier to a tool and **call the tool again**.

> 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
- TOOL TABLE: Press the **TOOL TABLE** soft key
- EDIT OFF ON: Press the **EDIT** soft key
- Move the cursor to the **KINEMATIC** column of the required tool
- SELECT: 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
- END: Exit the tool table

10.10 Dwell time FUNCTION DWELL

Programming dwell time

Application

The **FUNCTION DWELL** function enables you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

Procedure

Example

```
13 FUNCTION DWELL TIME10
```

Example

```
23 FUNCTION DWELL REV5.8
```

Proceed as follows for the definition:

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION DWELL

- **FUNCTION DWELL** soft key

DWELL TIME

- Press the **DWELL TIME** soft key
- Define the duration in seconds

DWELL REVOLUTIONS

- Alternatively, press the **DWELL REVOLUTIONS** soft key
- Define the number of spindle revolutions

11

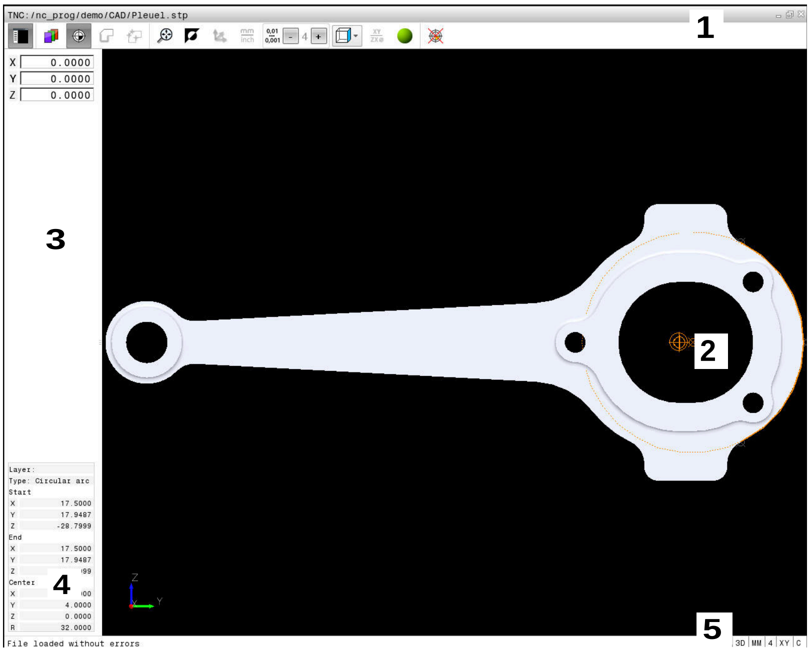
Data Transfer from CAD Files

11.1 Screen layout of the CAD viewer

Fundamentals of the CAD viewer

Screen display

When you open the **CAD-Viewer**, the following screen layout is displayed:



1 Menu bar
2 Graphics window
3 List View window
4 Window element information
5 Status bar

File formats

The **CAD-Viewer** enables you to open standardized CAD data formats directly on the control.

The control displays the following file formats:

File	Type	Format
Step	.STP and .STEP	■ AP 203 ■ AP 214
IGES	.IGS and .IGES	■ Version 5.3
DXF	.DXF	■ R10 to 2015

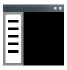
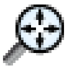
11.2 CAD viewer

Application

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

The preset can be positioned anywhere in the model. Starting from this preset, element information such as centers of circles can be shown. However, the control cannot execute it.

The following icons are available:

Icon	Setting
	Show or hide the Window List view to expand the Graphics window
	Display of the various layers
 	Set a preset or delete a set preset
	Set the zoom to the largest possible view of the complete graphics
	Change the background color (black or white)
0,01 ... 0,001	Set resolution: The resolution specifies how many decimal places the control will use when generating the contour program. Default setting: 4 decimal places with **mm** and 5 decimal places with **inch**
	Switch between various views of the model e.g. **Top**

> You can use icons to select contours and drilling positions, but the control cannot execute the elements.

12

Fundamentals / Overviews

12.1 Introduction

Frequently recurring machining cycles that comprise several working steps are stored in the control's memory as standard cycles. Coordinate transformations and several special functions are also available as cycles. Most cycles use Q parameters as transfer parameters.

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

- You should run a program test before machining

If you use indirect parameter assignments in cycles with numbers greater than 200 (e.g. **Q210 = Q1**), any change in the assigned parameter, e.g. in **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 machining cycles with numbers 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 control assigns the feed rate from the **TOOL CALL** block when processing the cycle definition.

If you want to delete a cycle that includes multiple sub-blocks, the control prompts you whether you want to delete the whole cycle.

12.2 Available cycle groups

Overview of fixed cycles

CYCL DEF

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

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring, tapping and counter-boring	314
POCKETS/ STUDS/ SLOTS	Cycles for milling , studs and slots and for face millingrec-tangular pockets and rectangu-lar studs	364
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	390
PATTERN	Cycles for producing point patterns	303
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop,	406

▷

- If required, switch to machine-specific fixed cycles. These fixed cycles can be integrated by your machine tool builder.

12.3 Working with fixed cycles

Machine-specific cycles

Cycles are available at many machines. Your machine manufacturer implements these cycles into the control, in addition to the HEIDENHAIN cycles. 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 functionality.

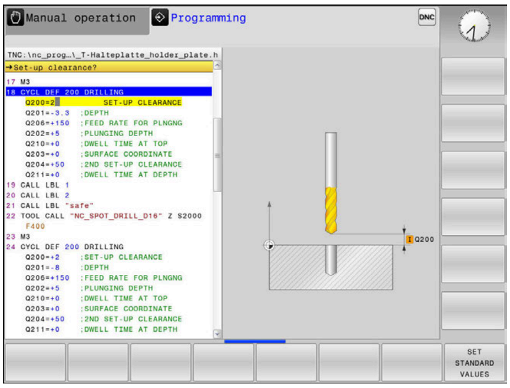
Some machine-specific cycles use transfer parameters that are also part of HEIDENHAIN standard cycles. In order to avoid problems (related to overwriting of transfer parameters that are used more than once), when using DEF-active cycles (cycles that the control runs automatically during cycle definition) and CALL-active cycles (cycles that you need to call to run them) used at the same time,

Further information: "Calling a cycle", Page 290

do the following: The following procedure is advisable:

- As a rule, always program DEF-active cycles before CALL-active cycles
- Only program a DEF-active cycle between the definition of a CALL-active cycle and the cycle call if there are no interferences of transfer parameters of these two cycles

Defining a cycle using soft keys

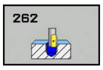


CYCL DEF

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

DRILLING/ THREAD

- Select the desired cycle group, e.g. drilling cycles



- Select the cycle, e.g. **DRILLING**. The control initiates a dialog and prompts you for all required input values. At the same time, a graphic is displayed in the right half of the screen.
- Enter all parameters required by the control. Conclude each input with the **ENT** key
- The control closes the dialog when all required data has been entered

Defining a cycle using the GOTO function

CYCL DEF

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

GOTO ☐

- The control displays a pop-up window with an overview of the cycles
- Select the desired cycle with the cursor keys or
- Enter the cycle number. Confirm each input with the **ENT** key. The control then initiates the cycle dialog as described above

Example

7 CYCL DEF 200 DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=3	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q210=0	;DWELL TIME AT TOP
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q211=0.25	;DWELL TIME AT DEPTH
Q395=0	;DEPTH REFERENCE

Calling a cycle

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Direction of rotation of the spindle (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 have been defined in the program. You cannot and must not call them:

- Cycle 220 for point patterns on circles and Cycle 221 for point patterns on lines
- Cycles for coordinate transformation
- Cycle 9 DWELL TIME
- All touch probe cycles

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

Calling a cycle with CYCL CALL

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

CYCL CALL

- ▶ To program the cycle call: Press the **CYCL CALL** key
- ▶ To enter the cycle call: Press the **CYCL CALL M** soft key
- ▶ If necessary, enter the M function (miscellaneous function) (such as **M3** to switch the spindle on), or close 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 point table.

Further information: "Pattern definition with PATTERN DEF", Page 296

Further information: "Point tables", Page 308

Calling a cycle with M89/M99

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

If the control is to execute the cycle automatically after every positioning block, program the first cycle call with **M89**.

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

- **M99** in the positioning block in which you move to the last starting point, or
- Use **CYCL DEF** to define a new fixed cycle

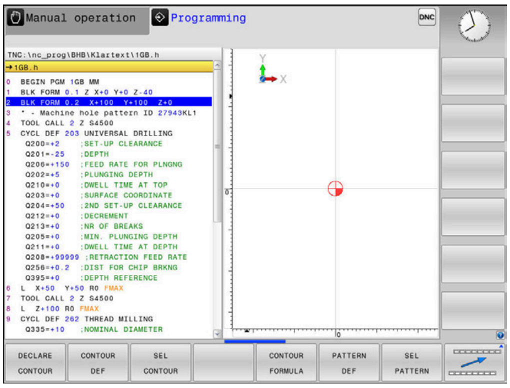
> In combination with FK programming, the control does not support M89!

12.4 Program defaults for cycles

Overview

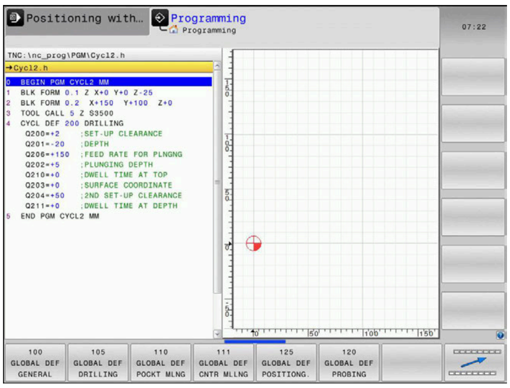
All Cycles 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** you can define these cycle parameters at the beginning of the program, so that they are effective globally for all machining cycles used in the NC program. In the respective machining cycle, you then simply reference 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	293
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	294
110 GLOBAL DEF POCKT MLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	294
111 GLOBAL DEF CNTR MLLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	294
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	295
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	295

Entering GLOBAL DEF



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

SPEC FCT

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

PROGRAM DEFAULTS

- Select the functions for program defaults

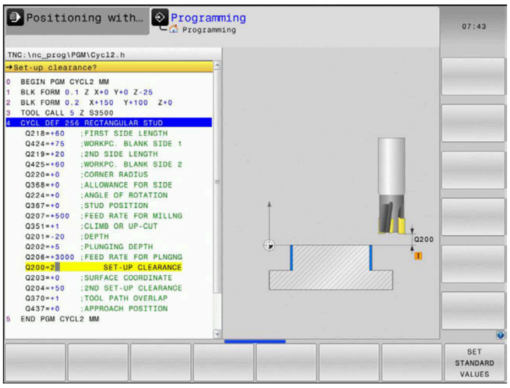
GLOBAL DEF

- Press the **GLOBAL DEF** soft key

100 GLOBAL DEF GENERAL

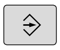
- Select the desired GLOBAL DEF function, e.g. by pressing the **GLOBAL DEF GENERAL** soft key
- Enter the required definitions, and confirm each entry with the **ENT** key

Using GLOBAL DEF information



If you entered the respective GLOBAL DEF functions at the start of the program, you can reference these globally valid values when defining any machining cycle.

Proceed as follows:



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

- Select machining cycles: Press the **CYCLE DEF** key

- Select the desired cycle group, e.g. drilling cycles

- Select the desired cycle, e.g. **drilling**
- If a global parameter exists, the control will display the **SET STANDARD VALUES** soft key

- Press the **SET STANDARD VALUES** soft key. The control enters the word **PREDEF** (predefined) in the cycle definition. This creates a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program

NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly.

- Make sure to use **GLOBAL DEF** carefully. Test your program before executing it
- If a fixed value is entered in machining cycles, **GLOBAL DEF** does not modify this value

Global data valid everywhere

- **Safety clearance**: Distance between tool face and workpiece surface for automated approach of the cycle start position in the tool axis
- **2nd set-up clearance:** Position to which the control positions the tool at the end of a machining step. The next machining position is approached at this height in the working plane
- **F positioning:** Feed rate at which the control traverses the tool within a cycle
- **F retraction:** Feed rate at which the control retracts the tool

The parameters are valid for all fixed cycles with numbers greater than 2xx.

Global data for drilling operations

- **Retraction rate for chip breaking:** Value by which the control 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 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

> For the TNC 128 straight-cut control, the **GLOBAL DEF CNTR MLLNG** soft key has no function. This soft key was added for reasons of compatibility.

Global data for positioning behavior

- **Positioning behavior**: Retraction in the tool axis at the end of a 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 control traverses the touch probe between measuring points if the **Move to clearance height** option is activated
- **Move to clearance height**: Select whether the control moves the touch probe to the set-up clearance or clearance height between the measuring points

> The parameters apply to all touch probe cycles numbered 4xx.

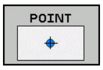
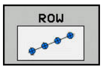
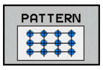
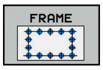
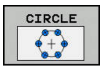
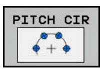
12.5 Pattern definition with PATTERN DEF

Application

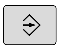
You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. Just like in cycle definitions, help graphics are available for pattern definition that clearly indicate the input parameters required.

> **NOTICE**
>
> **Danger of collision!**
>
> The **PATTERN DEF** function calculates the machining coordinates in the **X** and **Y** axes For all tools axes apart from **Z** there is a danger of collision in the following operation!
>
> - Use **PATTERN DEF** only in connection with the tool axis **Z**

The following machining patterns are available:

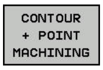
Soft key	Machining pattern	Page
	POINT Definition of up to any 9 machining positions	298
	ROW Definition of a single row, straight or rotated	298
	PATTERN Definition of a single pattern, straight, rotated or distorted	299
	FRAME Definition of a single frame, straight, rotated or distorted	300
	CIRCLE Definition of a full circle	301
	PITCH CIRCLE Definition of a pitch circle	302

Entering PATTERN DEF

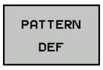


- Operating mode: Press the **Programming** key

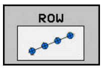
- Press the **SPEC FCT** key to select the special functions



- 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 290

The control then performs the most recently defined machining cycle based 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**: User's Manual for Setup, Testing and Running NC programs
>
> The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.
>
> If the coordinate surface in PATTERN DEF is larger than in the cycle, the 2nd set-up clearance references the coordinate surface in PATTERN DEF.
>
> If the coordinate surface in the cycle is larger than in PATTERN DEF, the set-up clearance references the sum of both coordinate surfaces.
>
> Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with Q352=1 (found under **SPEC FCT**/ Program Parameters). If you do so, the control will always position the tool at the second set-up clearance defined in the cycle.

Defining individual machining positions

You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.

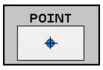
POS1 must be programmed with absolute coordinates. POS2 to POS9 can be programmed as absolute and/or incremental values.

If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

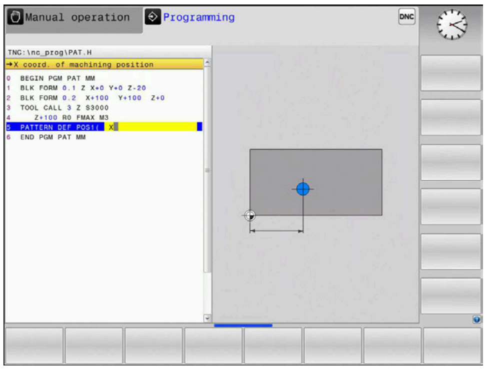
Example

```
10 Z+100 R0  FMAX
```

```
11 PATTERN DEF
   POS1 (X+25 Y+33.5  Z+0)
   POS2 (X+15 IY+6.5  Z+0)
```



- POS1: **X coord. of machining position** (absolute): Enter 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 the Y coordinate
- POS2: **Coordinate of workpiece surface** (absolute or incremental): Enter the Z coordinate



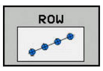
Defining a single row

If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

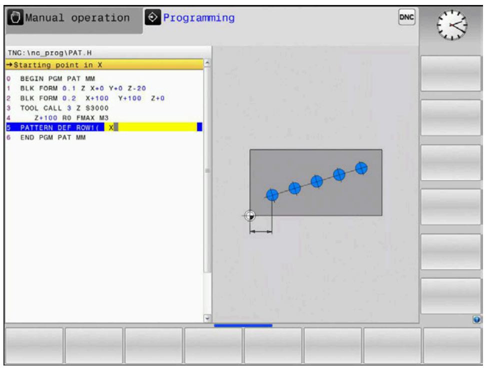
Example

```
10 Z+100 R0  FMAX
```

```
11 PATTERN DEF  ROW1
   (X+25 Y+33.5  D+8 NUM5  ROT+0 Z
   +0)
```

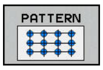


- **Starting point in X** (absolute): Coordinate of the pattern row starting point in the X axis
- **Starting point in Y** (absolute): Coordinate of the pattern row starting point in the Y axis
- **Spacing of machining positions** (incremental): Distance between the machining positions. You can enter a positive or negative value
- **Number of operations**: Total number of machining positions
- **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin



Defining a single pattern

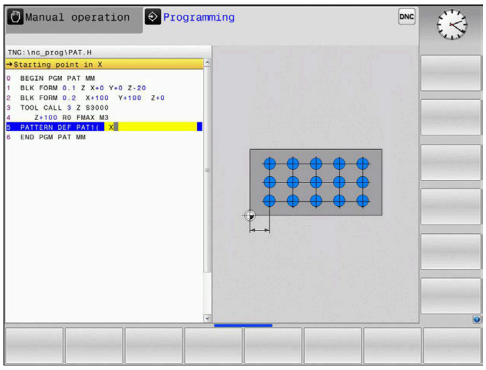
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.
>
> The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.



- **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- **Spacing of machining positions X** (incremental): Distance 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 about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation about which only the reference axis of the working 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 about which only the minor axis of the working 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

Example

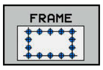
```
10 Z+100 R0 FMAX

11 PATTERN DEF  PAT1 (X+25 Y+33,5
   DX+8 DY+10  NUMX5 NUMY4 ROT+0
   ROTX+0 ROTY+0  Z+0)
```



Defining individual frames

If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

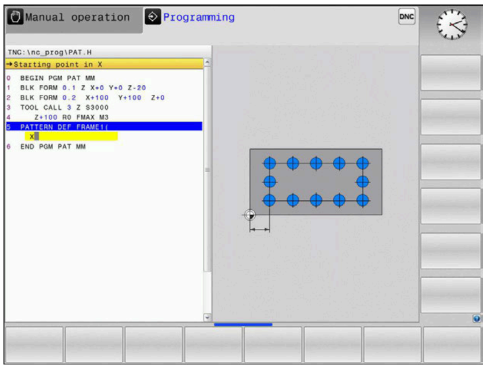
The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.



- **Starting point in X** (absolute): Coordinate of the frame starting point in the X axis
- **Starting point in Y** (absolute): Coordinate of the frame starting point 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 about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Rotary pos. ref. ax.**: Angle of rotation about which only the reference axis of the working 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 about which only the minor axis of the working 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

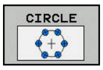
Example

```
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

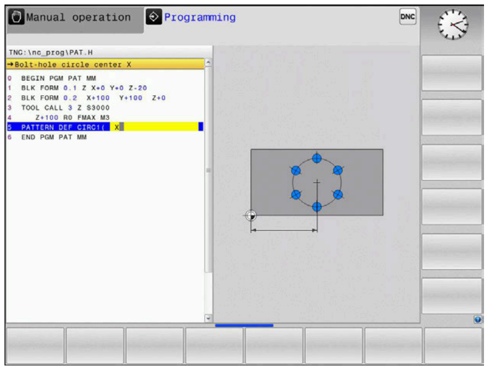
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **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 circular hole pattern
- **Starting angle**: Polar angle of the first machining position. Reference axis: Principal axis of the active working 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

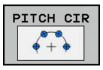
Example

```
10 Z+100 R0 FMAX
11 PATTERN DEF CIRC1
    (X+25 Y+33 D80 START+45 NUM8 Z
    +0)
```



Defining a pitch circle

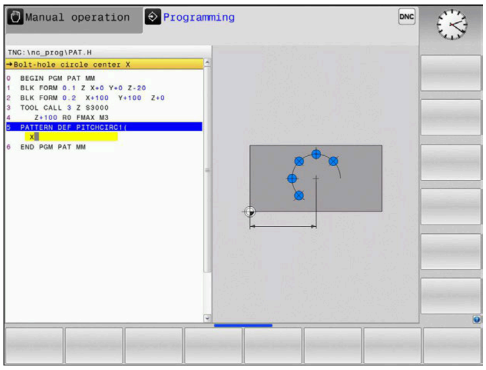
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Bolt-hole circle center X** (absolute): Coordinate of the 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 circular hole pattern
- **Starting angle**: Polar angle of the first machining position. Reference axis: Principal axis of the active working 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

Example

```
10 Z+100 R0  FMAX
11 PATTERN DEF  PITCHCIRC1
    (X+25  Y+33 D80  START+45 STEP30
    NUM8 Z+0)
```



12.6 POLAR PATTERN (Cycle 220)

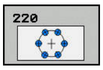
Cycle run

1. The control moves the tool at rapid traverse 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 control executes the last defined fixed machining cycle
3. The tool then approaches the starting point for the next machining operation on a straight line. 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.

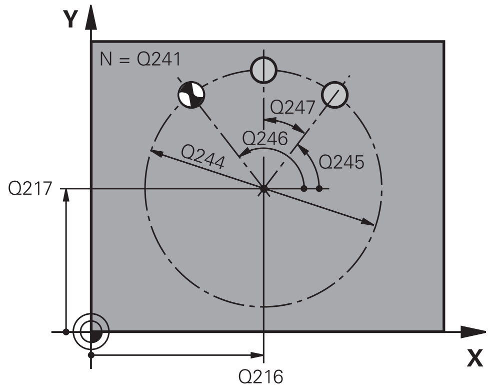
Please note while programming:

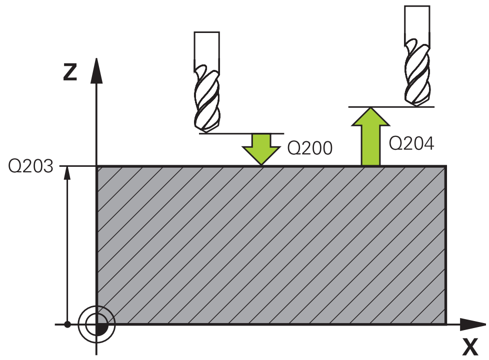
> Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.
>
> If you combine Cycle 220 with one of the fixed cycles 200 to 207 and 251, 253 and 256, the set-up clearance, workpiece surface and the 2nd set-up clearance that were defined in Cycle 220 or 221 will be effective. This applies within the NC program until the affected parameters are overwritten again. Example: If, in an NC program, Cycle 200 is defined with Q203=0 and you then program a Cycle 220 with Q203=-5, then the subsequent call with CYCL CALL and M99 will use Q203=-5. Cycles 220 and 221 overwrite the above-mentioned parameters of CALL-active machining cycles (if the same input parameters have been programmed in both cycles).
>
> If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q216 Center in 1st axis?** (absolute): Pitch circle center in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q217 Center in 2nd axis?** (absolute): Pitch circle center in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q244 Pitch circle diameter?**: Diameter of the pitch circle. Input range: 0 to 99999.9999
- **Q245 Starting angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range: -360.000 to 360.000
- **Q246 Stopping angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you specify a 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 a stepping angle of 0, the control will calculate the stepping angle from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the control will not take the stopping angle into account. The sign for the stepping angle 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





Example

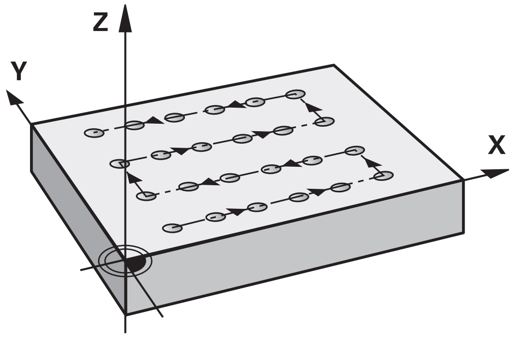
53 CYCL DEF 220 POLAR PATTERN	
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q244=80	;PITCH CIRCLE DIAMETR
Q245=+0	;STARTING ANGLE
Q246=+360	;STOPPING ANGLE
Q247=+0	;STEPPING ANGLE
Q241=8	;NR OF REPETITIONS
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE

- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the tool is to move between machining operations:
  0: Move to set-up clearance between machining operations
  1: Move to 2nd set-up clearance between machining operations

Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE

12.7 LINEAR POINT PATTERN (Cycle 221)

Cycle run



1 The control 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 control executes the last defined fixed machining cycle
3 Then, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis. The tool stops at the set-up clearance (or the 2nd set-up clearance)
4 This procedure (steps 1 to 3) will be repeated until all machining operations from the first line have been completed. The tool is located above the last point of 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 direction of the reference axis.
7 This process (6) is repeated until all machining operations in the second line have been executed.
8 The tool then moves to the starting point of the next line
9 All subsequent lines are processed in a reciprocating movement.

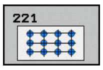
Please note while programming:

Cycle 221 is DEF active, which means that Cycle 221 automatically calls the last defined fixed cycle.

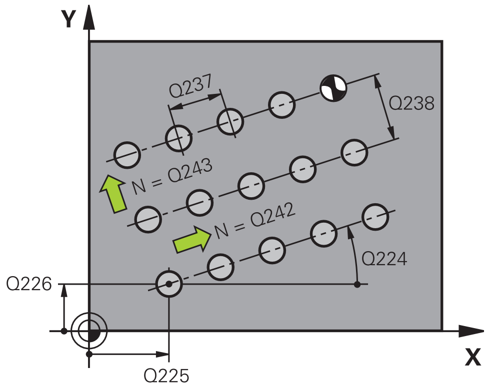
If you combine Cycle 221 with one of the fixed cycles 200 to 207 and 251, 253 and 256, the set-up clearance, workpiece surface, the 2nd set-up clearance, and the rotational position that were defined in Cycle 221 will be effective.

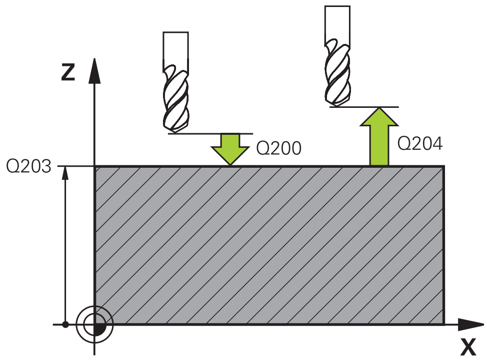
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q225 Starting point in 1st axis?** (absolute): Coordinate of the starting point in the reference axis of the working plane
- **Q226 Starting point in 2nd axis?** (absolute): Coordinate of the starting point in the minor axis of the working plane
- **Q237 Spacing in 1st axis?** (incremental): Spacing between the individual points on the 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 is located at the starting point
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the tool is to move between machining operations:
  0: Move to set-up clearance between machining operations
  1: Move to 2nd set-up clearance between machining operations





Example

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

12.8 Point tables

Application

You should create a point table whenever you want to run a cycle or several cycles in sequence, to machine 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.

Entering values into a point table

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

PGM MGT

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

FILE NAME?

ENT

- Enter the name and file type of the point table. Confirm with the **ENT** key

MM

- Select the unit of measure: Press the **MM** or **INCH** soft key. The control changes to the program window and displays an empty points table

INSERT LINE

- Press the **INSERT LINE** soft key to insert a new line. 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 **SORT/ HIDE COLUMNS** soft keys (fourth soft-key row) to specify which coordinates you want to enter into 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

ENT

- Activate hiding or

NO ENT

- Deactivate hiding

Selecting a point table in the NC program

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

PGM CALL

- Press the **PGM CALL** key to call the function for selecting the point table

SELECT POINT TABLE

- Press the **SELECT POINT TABLE** soft key

SELECT FILE

- Press the **SELECT FILE** soft key
- Select the point table and confirm with the **OK** soft key

If the point table is not stored in the same directory as the NC program, you must enter the complete path.

Example

```
7 SEL PATTERN  "TNC:\DIRKT5\NUST35.PNT"
```

Calling a cycle in connection with point tables

If you want the control to call the cycle at the points that you last defined in a point table, then program the cycle call with **CYCLE CALL PAT**:

CYCL CALL

- To program the cycle call: Press the **CYCL CALL** key
- To call the point table, press the **CYCL CALL PAT** soft key
- Enter the feed rate at which the control is to move from point to point or press the **F MAX** soft key (if you make no entry, the control will move at the last programmed feed rate)
- Enter a miscellaneous function (M function) if required. Confirm your input with the **END** key

The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.

Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with Q352=1 (found under **SPEC FCT**/Program Parameters). If you do so, the control will always position the tool at the second set-up clearance defined in the cycle.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the M103 miscellaneous function.

Effect of the point table with Cycles 200 to 207

The control 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 as the starting point coordinate in the spindle axis, you must define the coordinate of the workpiece upper edge (Q203) as 0.

Effect of the point table with Cycles 251, 253 and 256

The control interprets the points on the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table as the starting point coordinate in the spindle axis, you must define the coordinate of the workpiece upper edge (Q203) as 0.

> If you call **CYCL CALL PAT**, the control will use the point table that you defined last. This is also the case if you defined the point table in an NC program nested with **CALL PGM**.

NOTICE

Danger of collision!

If you program a clearance height for any points in a point table, the control will ignore the 2nd set-up clearance for **all** points of this machining cycle!

- Program GLOBAL DEF 125 POSITIONING beforehand. This will ensure that the control considers the clearance height from the point table for the corresponding point only.

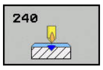
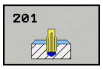
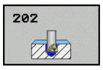
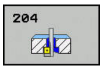
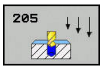
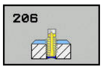
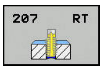
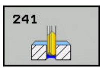
13

Cycles: Drilling cycles / thread cycles

13.1 Fundamentals

Overview

The control provides 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	315
	200 DRILLING With automatic pre-positioning, 2nd set-up clearance	317
	201 REAMING With automatic pre-positioning, 2nd set-up clearance	319
	202 BORING With automatic pre-positioning, 2nd set-up clearance	321
	203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	324
	204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	330
	205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	334
	206 TAPPING With floating tap holder, 2nd set-up clearance, dwell time at depth	353
	207 RIGID TAPPING With thread depth and thread pitch	356
	241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition	342

13.2 CENTERING (Cycle 240)

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified 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 is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with the radius compensation **R0**

The algebraic sign for the **Q344** (diameter) or **Q201** (depth) cycle parameter determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

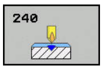
NOTICE

Danger of collision!

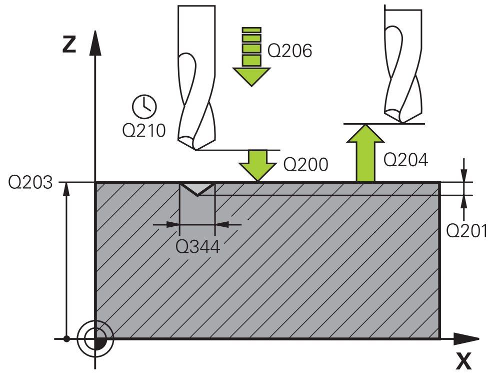
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

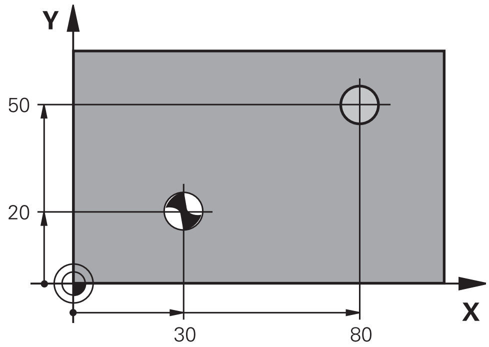
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range: 0 to 99999.9999
- **Q343 Select diameter/depth (1/0)**: Select whether centering is based on the entered diameter or depth. If the control 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.T tool table.
  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





Example

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

13.3 DRILLING (Cycle 200)

Cycle run

1. The control positions the tool in the spindle 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 programmed feed rate **F**
3. The Control retracts 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 control repeats this process (2 to 4) until the programmed depth is reached (the dwell time from Q211 is effective with every infeed)
6. Finally, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

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

If you want to drill without chip breaking, make sure to define, in the **Q202** parameter, a higher value than the depth **Q201** plus the calculated depth based on the point angle. You can enter a much higher value there.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

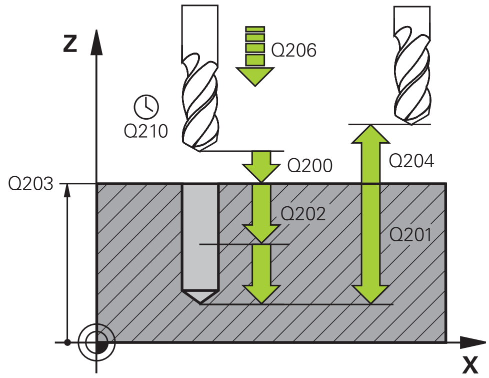
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

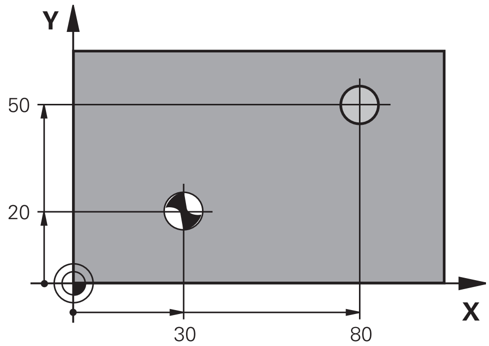
Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min 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 control will go to depth in one movement if:

  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal by the Control. 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 control 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.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool





Example

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	

13.4 REAMING (Cycle 201)

Cycle run

1 The control positions the tool in the spindle 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 Then, the control retracts the tool at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

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

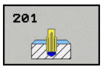
NOTICE

Danger of collision!

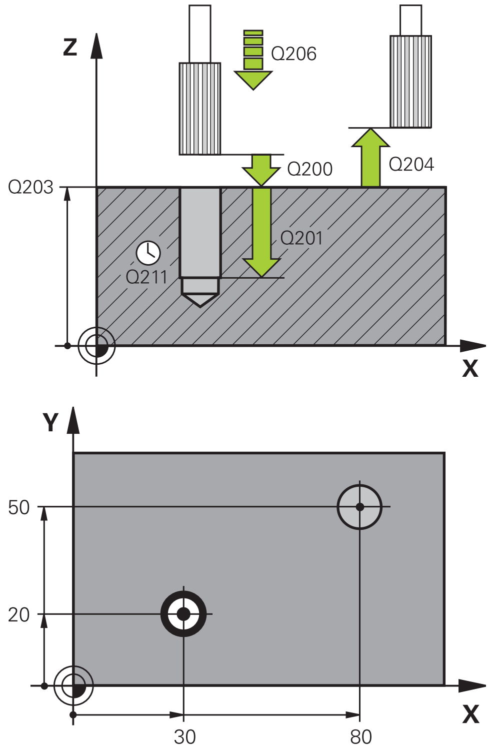
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min 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



Example

```
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
```

13.5 BORING (Cycle 202)

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified 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 control then carries out an oriented spindle stop to the position that is defined in the **Q336** parameter
5. If retraction is selected, the control 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**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**. If **Q214**=0 the tool tip remains on the wall of the hole
7. The control then returns the tool to the center of the hole

Please note while programming:

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

This cycle can only be used on machines with a 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 DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

After machining, the control returns the tool to the starting point of the working plane. This way, you can continue positioning the tool incrementally.

If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

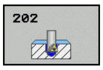
NOTICE

Danger of collision!

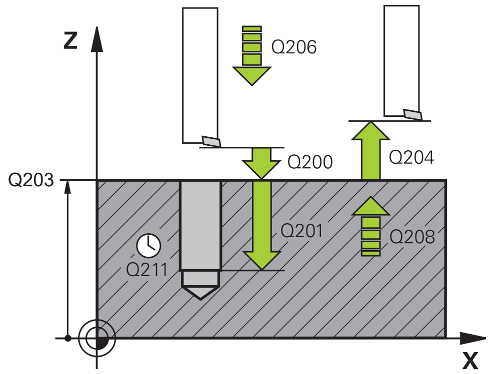
There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

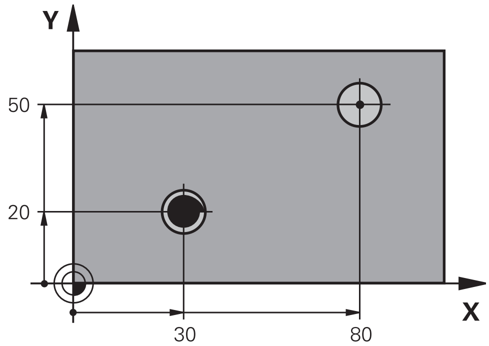
- Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min 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 control retracts the tool at the hole bottom (after carrying out an oriented spindle stop)
  0: Do not disengage the tool
  1: Disengage the tool in the minus direction of the reference axis
  2: Disengage the tool in the minus direction of the minor axis
  3: Disengage the tool in the plus direction of the reference axis
  4: Disengage the tool in the plus direction of the minor axis
- **Q336 Angle for spindle orientation?** (absolute): Angle to which the control positions the tool before retracting it. Input range: -360.000 to 360.000





Example

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

13.6 UNIVERSAL DRILLING (Cycle 203)

Cycle run

Behavior without chip breaking and without decrement:

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
2. The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3. Then, the control retracts the tool from the hole to **SET-UP CLEARANCE Q200**
4. Now, the control again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTHQ202 FEED RATE FOR PLNGNG Q206**
5. When machining without chip breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and remains there for the **DWELL TIME AT TOP Q210**.
6. This procedure is repeated until **depth Q201** is achieved.
7. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and without decrement:

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
2. The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3. Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4. Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** at **FEED RATE FOR PLNGNG Q206**
5. The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
6. If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**
7. Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8. Steps 2 to 7 will be repeated until **DEPTH Q201** is reached.
9. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and with decrement

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified **SAFETY CLEARANCE Q200** above the workpiece surface
2. The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3. Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4. Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** minus **DECREMENT Q212** at **FEED RATE FOR PLNGNG Q206**. The increasingly smaller difference between the updated **PLUNGING DEPTH Q202** minus **DECREMENT Q212** must never be smaller than **MIN. PLUNGING DEPTH Q205** (example: **Q202**=5, **Q212**=1, **Q213**=4, **Q205**= 3: The first plunging depth is 5 mm, the second plunging depth is 5 - 1 = 4 mm, the third plunging depth is 4 - 1 = 3 mm, the fourth plunging depth is also 3 mm)
5. The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
6. If programmed, the control will now wait for the time specified in **DWELL TIME AT TOP Q210**
7. Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8. Steps 2 to 7 will be repeated until **DEPTH Q201** is reached.
9. If programmed, the control will now wait for the time specified in **DWELL TIME AT DEPTH Q211**
10. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

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

NOTICE

Danger of collision!

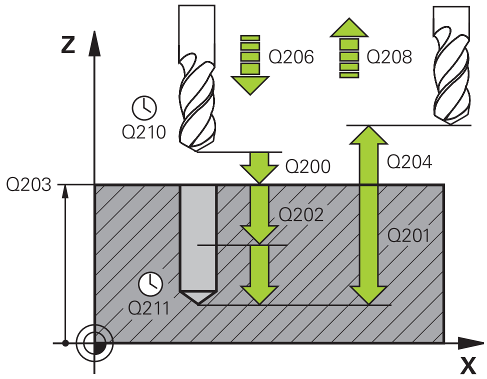
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min 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 control will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal by the Control. 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 control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999
- **Q213 Nr of breaks before retracting?**: Number of chip breaks before the control will retract the tool from the hole for chip removal. For chip breaking, the control 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 control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999



Example

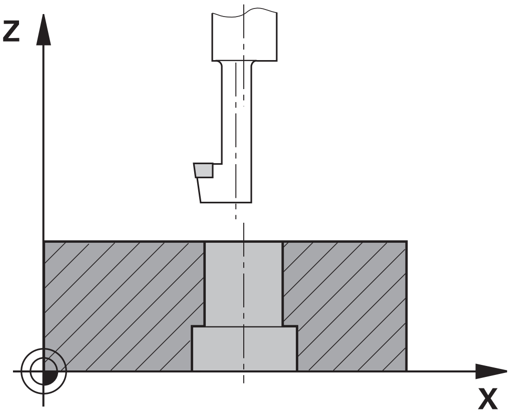
11 CYCL DEF 203 UNIVERSAL DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q211=0	;DWELL TIME AT TOP
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q212=0.2	;DECREMENT
Q213=3	;NR OF BREAKS
Q205=3	;MIN. PLUNGING DEPTH
Q211=0.25	;DWELL TIME AT DEPTH
Q208=500	;RETRACTION FEED RATE
Q256=0.2	;DIST FOR CHIP BRKNG
Q395=0	;DEPTH REFERENCE

- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the control retracts the tool at the feed rate specified in **Q206**. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control 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 control 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.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool

13.7 BACK BORING (Cycle 204)

Cycle run

This cycle allows counterbores to be machined from the underside of the workpiece.



1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface
2. The control 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 cutting edge has reached programmed set-up clearance beneath the lower workpiece edge
4. The control then centers the tool again in the bore hole, switches on the spindle and the coolant and moves at the feed rate for counterboring to the depth programmed for the counterbore
5. If programmed, the tool remains at the counterbore bottom. The tool will then be retracted from the hole again. The control carries out another oriented spindle stop and the tool is once again displaced by the off-center distance
6. Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**
7. The control then returns the tool to the center of the hole

Please note while programming:

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

This cycle can only be used on machines with a closed-loop 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 control returns the tool to the starting point of the working plane. This way, you can continue positioning the tool incrementally.

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

Enter the tool length so that the lower edge of the boring bar is measured, not the cutting edge.

When calculating the starting point for boring, the control considers the cutting edge length of the boring bar and the thickness of the material.

If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

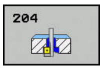
NOTICE

Danger of collision!

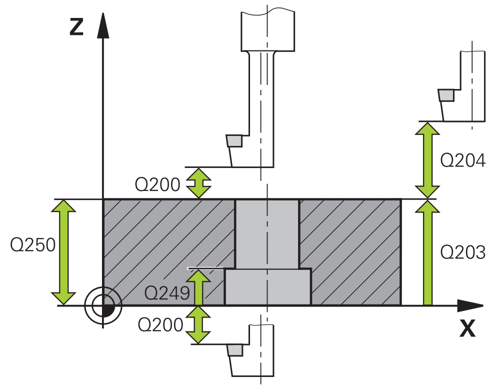
There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

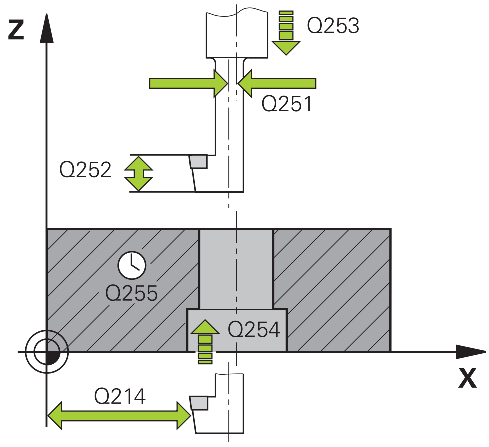
- Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q249 Depth of counterbore?** (incremental): Distance between underside of workpiece and bottom of the 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 edge; value from tool data sheet. Input range: 0.0001 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Q255 Dwell time in secs.?**: Dwell time in seconds at the bottom 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





Example

11 CYCL DEF 204 BACK BORING	
Q200=2	;SET-UP CLEARANCE
Q249=+5	;DEPTH OF COUNTERBORE
Q250=20	;MATERIAL THICKNESS
Q251=3.5	;OFF-CENTER DISTANCE
Q252=15	;TOOL EDGE HEIGHT
Q253=750	;F PRE-POSITIONING

- **Q214 Disengaging directn (0/1/2/3/4)?**: Determine the direction in which the control will displace the tool by the off-center distance (after having carried out an oriented spindle stop); programming 0 is not allowed
  1: Retract the tool in the negative direction of the reference axis
  2: Retract the tool in the negative direction of the minor axis
  3: Retract the tool in the positive direction of the reference axis
  4: Retract the tool in the positive direction of the minor axis
- **Q336 Angle for spindle orientation?** (absolute): Angle at which the control positions the tool before it is plunged into or retracted from the bore hole. Input range: -360.0000 to 360.0000

Q254=200	;F COUNTERBORING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE

13.8 UNIVERSAL PECKING (Cycle 205)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 If you enter a deepened starting point, the control moves at the defined positioning feed rate to the set-up clearance above the recessed starting point
3 The tool drills to the first plunging depth at the programmed feed rate **F**
4 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered advanced stop distance above the first plunging depth
5 The tool then drills deeper by the plunging depth at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
6 The control repeats this procedure (steps 2 to 4) until the 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 or the 2nd set-up clearance at the retraction feed rate. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

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

If you enter advance stop distances **Q258** not equal to **Q259**, the control will change the advance stop distances between the first and last plunging depths at the same rate.

If you use **Q379** to enter a recessed starting point, the control will change the starting point of the infeed movement. Retraction movements are not changed by the control, they are always calculated with respect to the coordinate of the workpiece surface.

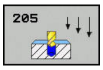
NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

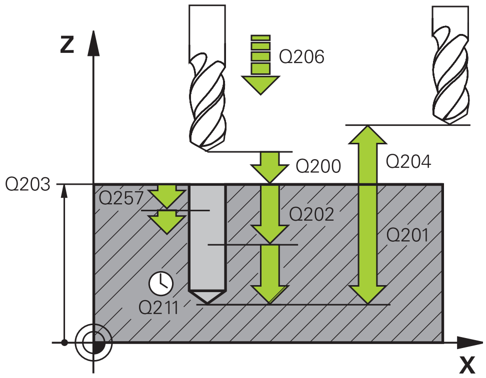
Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range: -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min 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 control 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 control decreases the **Q202** plunging depth. Input range: 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999
- **Q258 Upper advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole. Input range 0 to 99999.9999
- **Q259 Lower advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole; value for the last plunging depth. Input range: 0 to 99999.9999



Example

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

- **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control 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 control retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **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, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed 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). Input 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 control retracts the tool at the feed rate specified in **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 control 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.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool

Position behavior when working with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Start of drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,2 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2
- The starting point of drilling is calculated as follows: 0.2 x Q379=0.2*2=0.4; the starting point is 0.4 mm/inch above the recessed starting point. So if the recessed starting point is at -2, the control starts the drilling process at -1.6 mm

The following table shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.2 * Q379	Start of drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (Q200=2, 5>2, thus the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (Q200=2, 20>2, thus the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (Q200=5, 20>5, thus the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip breaking

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,8 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2
- The position for chip removal is calculated as follows: 0.8 x Q379 = 0.8 * 2 = 1.6; the position for chip removal is 1.6 mm/inch above the recessed start point. So if the recessed starting point is at -2, the control starts chip removal at -0.4

The following table shows various examples for calculating the position for chip breaking (retraction position):

Position for chip breaking (retraction position) with deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (Q200=2, 8>2, thus the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (Q200=2, 20>2, thus the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (Q200=2, 80>2, thus the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (Q200=5, 8>5, thus the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (Q200=5, 20>5, thus the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (Q200=5, 80>5, thus the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (Q200=20, 80>20, thus the value 20 is used.)	-80

13.9 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **Safety clearance Q200** above the workpiece **SURFACE COORDINATE Q203**
2. Depending on the "Position behavior when working with Q379", Page 338, the control will either switch on the spindle with the programmed speed at the **Safety clearance Q200** or at a certain distance above the coordinate surface. see Page 338
3. The control executes the approach motion depending on the direction of rotation defined in the cycle with a spindle that rotates clockwise, counterclockwise, or is stationary
4. The tool drills to the hole depth at the feed rate **F**, or to the maximum 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 control reduces the feed rate by the feed rate factor after the dwell depth has been reached
5. If programmed, the tool remains at the hole bottom for chip breaking.
6. The control repeats this procedure (steps 4 to 5) until the total hole depth is reached
7. After the control has reached this position, it will automatically switch off the coolant as soon as the speed has reached the value defined in Q427 **ROT.SPEED INFEED/OUT**
8. The control positions the tool to the retract position at the retraction feed rate. To find out the retract position value in your particular case, please refer to: see Page 338
9. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

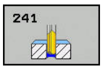
NOTICE

Danger of collision!

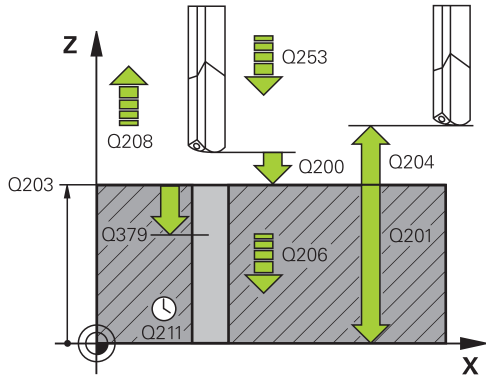
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and **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 workpiece datum. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q379 Deepened starting point?** (incremental, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed 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). Input 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 control 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
- **Q428 Spindle speed for drilling?**: Desired speed for drilling. Input range: 0 to 99999



Example

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

- **Q429 M function for coolant on?**: Miscellaneous function for switching on the coolant. The control switches on 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 control 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 (default setting). Application: During machining of through-holes some tools require a short dwell time before leaving 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 control 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 control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999

Position behavior when working with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Start of drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,2 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2
- The starting point of drilling is calculated as follows: 0.2 x Q379=0.2*2=0.4; the starting point is 0.4 mm/inch above the recessed starting point. So if the recessed starting point is at -2, the control starts the drilling process at -1.6 mm

The following table shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.2 * Q379	Start of drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (Q200=2, 5>2, thus the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (Q200=2, 20>2, thus the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (Q200=5, 20>5, thus the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip breaking

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,8 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2
- The position for chip removal is calculated as follows: 0.8 x Q379 = 0.8 * 2 = 1.6; the position for chip removal is 1.6 mm/inch above the recessed start point. So if the recessed starting point is at -2, the control starts chip removal at -0.4

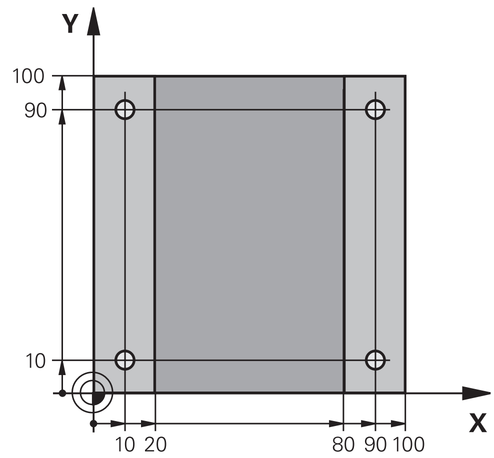
The following table shows various examples for calculating the position for chip breaking (retraction position):

Position for chip breaking (retraction position) with deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (Q200=2, 8>2, thus the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (Q200=2, 20>2, thus the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (Q200=2, 80>2, thus the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (Q200=5, 8>5, thus the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (Q200=5, 20>5, thus the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (Q200=5, 80>5, thus the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (Q200=20, 80>20, thus the value 20 is used.)	-80

13.10 Programming Examples

Example: Drilling cycles



0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
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, cycle call
8 X+90 R0 FMAX M99	Approach hole 2, cycle call
9 Y+90 R0 FMAX M99	Approach hole 3, cycle call
10 X+10 R0 FMAX M99	Approach hole 4, cycle call
11 Z+250 R0 FMAX M2	Retract the tool, end program
12 END PGM C200 MM	

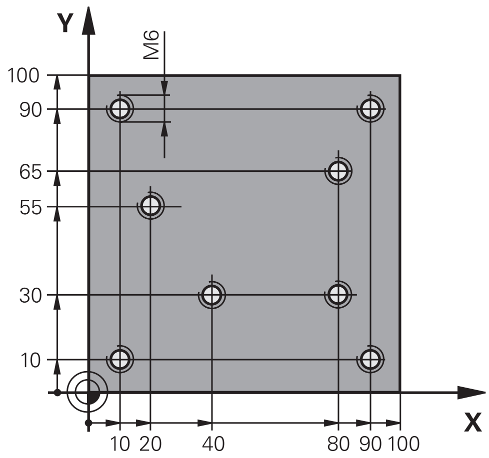
Example: Using drilling cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the pattern definition PATTERN DEF POS and are called by the Control with CYCL CALL PAT.

The tool radii have been selected in such a way 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)
  Further information: "Fundamentals", Page 314



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL CALL 1 Z S5000	Tool call: centering tool (tool radius 4)
4 Z+50 R0 FMAX	Move tool to clearance height
5 PATTERN DEF	Define all drilling positions in the point pattern
POS1(X+10 Y+10 Z+0)	
POS2(X+40 Y+30 Z+0)	
POS3(X+20 Y+55 Z+0)	
POS4(X+10 Y+90 Z+0)	
POS5(X+90 Y+90 Z+0)	
POS6(X+80 Y+65 Z+0)	
POS7(X+80 Y+30 Z+0)	
POS8(X+90 Y+10 Z+0)	
6 CYCL DEF 240 CENTERING	Cycle definition: centering
Q200=2 ;SET-UP CLEARANCE	
Q343=0 ;SELECT DIA./DEPTH	
Q201=-2 ;DEPTH	
Q344=-10 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
POSITION 7 GLOBAL DEF 125	This function is used for CYCL CALL PAT and positions the tool at the 2nd set-up clearance between the points. This function remains active until M30 is executed.
Q345=+1 ;SELECT POS. HEIGHT	
7 CYCL CALL PAT F5000 M13	Cycle call in connection with the point pattern

8 Z+100 R0 FMAX	Retract the tool
9 TOOL CALL 2 Z S5000	Tool call: drill (radius 2.4)
10 Z+50 R0 F5000	Move tool to clearance height
11 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
12 CYCL CALL PAT F500 M13	Cycle call in connection with the point pattern
13 Z+100 R0 FMAX	Retract the tool
14 TOOL CALL Z S200	Tool call: tap (radius 3)
15 Z+50 R0 FMAX	Move tool to clearance height
16 CYCL DEF 206 TAPPING	Cycle definition: tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
17 CYCL CALL PAT F5000 M13	Cycle call in connection with the point pattern
18 Z+100 R0 FMAX M2	Retract the tool, end program
19 END PGM 1 MM	

13.11 TAPPING with a floating tap holder (Cycle 206)

Cycle run

1 The control positions the tool in the spindle 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 DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

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

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (feed rate override is not active). The Control then adapts the spindle speed as required.
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

The spindle speed potentiometer is inactive.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message. In Cycle 206, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

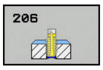
NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

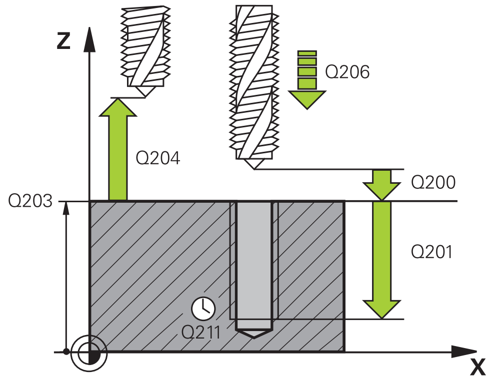
Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

Guide value: 4x pitch.

- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the 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



Example

25 CYCL DEF 206 TAPPING NEU	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q206=150	;FEED RATE FOR PLNGNG
Q211=0.25	;DWELL TIME AT DEPTH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

The feed rate is calculated as follows: F = S x p

F: Feed rate (mm/min)

S: Spindle speed (rpm)

p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the **NC Stop** key, the control will display a soft key with which you can retract the tool.

13.12 TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207)

Cycle run

The control cuts the thread without a floating tap holder in one or more passes.

1 The control positions the tool in the spindle 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 and the tool is retracted to the set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**
4 The control stops the spindle turning at that set-up clearance

Please note while programming:

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

This cycle is effective only for machines with 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 DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (speed override is not active). The Control then adapts the spindle speed as required.
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
- **limitSpindleSpeed** (no. 113604): Spindle speed limit
  True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time
  False: (Limiting not active)

The spindle speed potentiometer is inactive.

If you program M3 (or M4) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the TOOL CALL block).

If you do not program M3 (or M4) before this cycle, the spindle stands still after the end of the cycle. Then you must restart the spindle with M3 (or M4) before the next operation.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.

For tapping, the spindle and the tool axis are always synchronized with each other. The synchronization can be carried out while the spindle is rotating or while it is stationary.

If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis leaves the acceleration path within this distance.

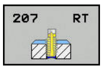
NOTICE

Danger of collision!

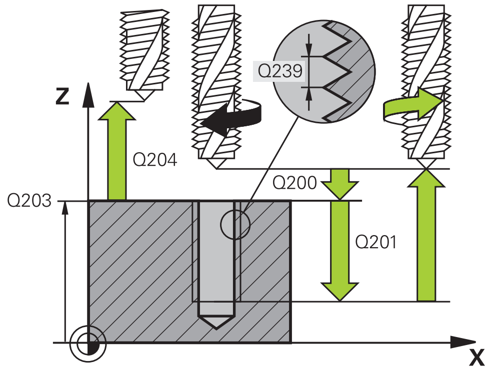
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the 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



Example

26 CYCL DEF 207 RIGID TAPPING NEU	
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 lower soft-key row. 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. The control 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 control displays the **MANUAL TRAVERSE** soft key. After you pressed the **MANUAL TRAVERSE** soft key, you can retract the tool in the active spindle axis. To resume machining after the interruption, press the **RESTORE POSITION** soft key and **NC Start**. The control moves the tool back to the position it had assumed before the **NC Stop** key was pressed.

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

13.13 Programming Examples

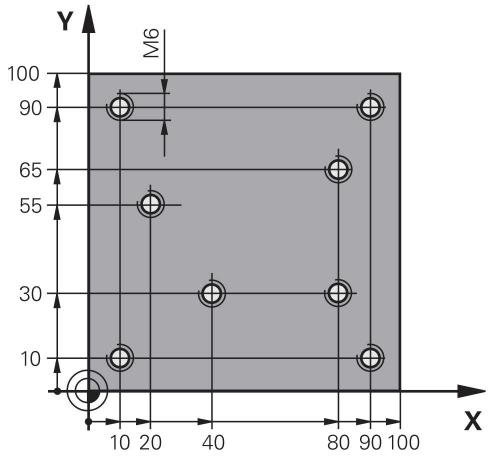
Example: Thread milling

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the control with **CYCL CALL PAT**.

The tool radii have been selected in such a way 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	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Tool call: centering tool
4 Z+10 R0 F5000	Move tool to clearance height (program a value for F): the control positions the tool at the clearance height after every cycle
5 SEL PATTERN “TAB1“	Select the 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
12 TOOL CALL 2 Z S5000	Tool call: drill
13 Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	

Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
15 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
16 Z+100 R0 FMAX M6	Retract the tool
17 TOOL CALL 3 Z S200	Tool call: tap
18 Z+50 R0 FMAX	Move tool to clearance height
19 CYCL DEF 206 TAPPING	Cycle definition: 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	

TAB1. PNT point table

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]

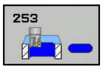
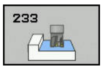
14

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling

14.1 Fundamentals

Overview

The control 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	365
	253 SLOT MILLING Roughing/finishing cycle with selection of machining operation	370
	256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	374
	233 FACE MILLING Machining the face with up to 3 limits	378

14.2 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 control 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 control moves the tool away from the pocket wall, then moves to 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 control plunges and then approaches the contour. The control first finishes the pocket walls, with multiple infeeds, if so specified.
6. Then the control finishes the floor of the pocket from the inside out.

Please note while programming!

Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

At the end of the cycle, the control returns the tool to the starting position.

At the end of a roughing operation, the control returns the tool to the pocket center at rapid traverse. The tool is positioned at set-up clearance above the current plunging depth. Program a sufficient set-up clearance so that the tool cannot jam because of chips.

At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

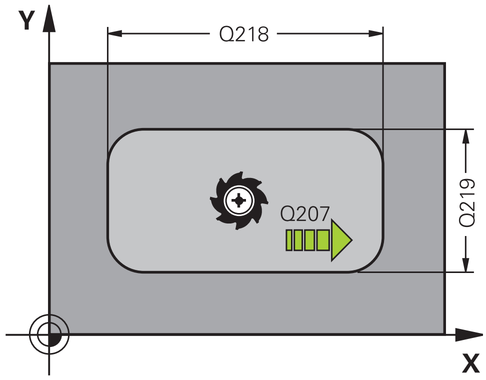
NOTICE

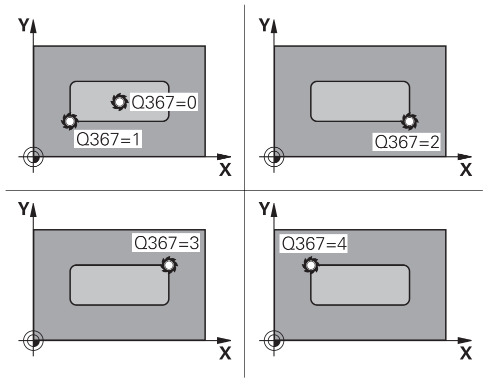
Danger of collision!

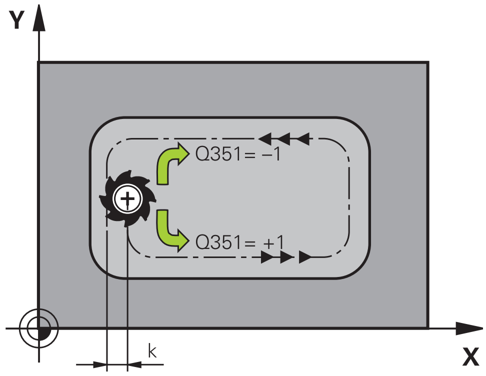
If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

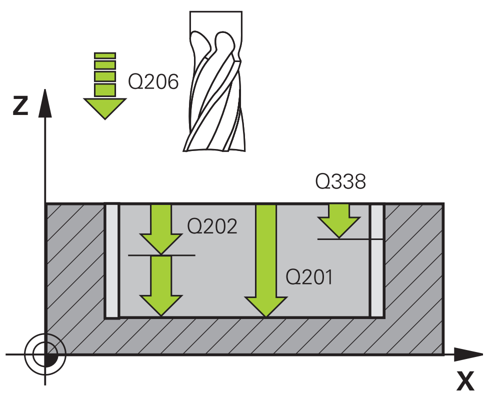
- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

Cycle parameters



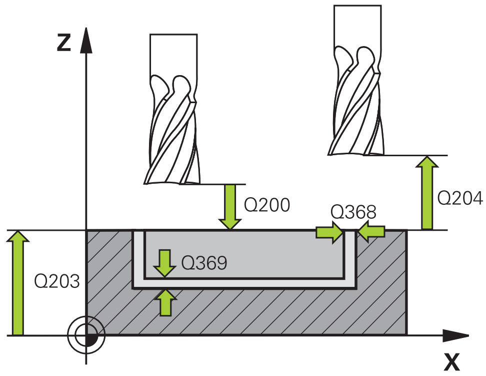






- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q218 First side length?** (incremental): 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 with respect to the position of the tool when the cycle is called:
  0: Tool position = pocket center
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner
- **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 working plane. Input range 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999;

- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999;
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (if you enter 0, climb milling is performed)
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1,9999



Example

8 CYCL DEF 251 RECTANGULAR POCKET
Q215=0 ;MACHINING OPERATION
Q218=80 ;FIRST SIDE LENGTH
Q219=60 ;2ND SIDE LENGTH
Q201=-20 ;DEPTH
Q367=0 ;POCKET POSITION
Q202=5 ;PLUNGING DEPTH
Q207=500 ;FEED RATE FOR MILLNG
Q206=150 ;FEED RATE FOR PLNGNG
Q385=500 ;FINISHING FEED RATE
Q368=0.2 ;ALLOWANCE FOR SIDE
Q369=0.1 ;ALLOWANCE FOR FLOOR
Q338=5 ;INFEED FOR FINISHING
Q200=2 ;SET-UP CLEARANCE
Q203=+0 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q351=+1 ;CLIMB OR UP-CUT
Q370=1 ;TOOL PATH OVERLAP
9 X+50 R0 FMAX
10 Y+50 R0 FMAX M3 M99

14.3 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 at the FEED RATE FOR PLUNGING Q206 to the first PLUNGING DEPTH Q202. The slot created by the roughing process is exactly as wide as the diameter of the tool. During the roughing process, 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 to the set-up clearance Q200 and then returns to the starting position from where it is moved 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 control 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 If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially in the left slot arc
7 Then the control finishes the floor of the slot from the inside out.

Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start!

- Do **not** program any incremental dimensions after this cycle
- Program an absolute position in all main axes after this cycle

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

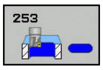
NOTICE

Danger of collision!

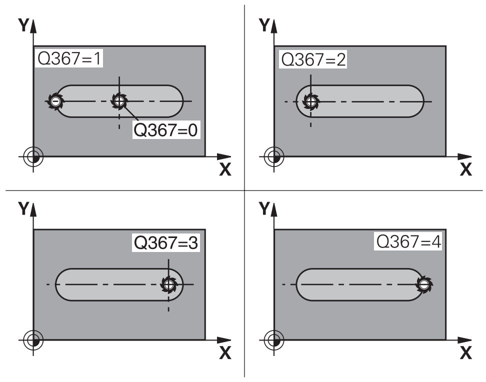
After roughing, the slot width equals the tool diameter, regardless of parameter Q219!

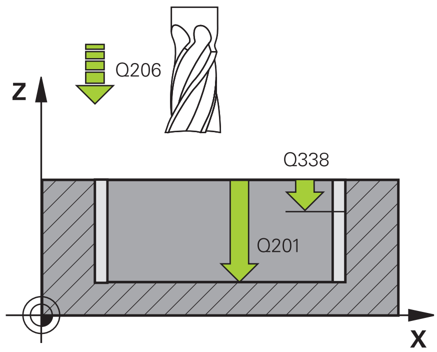
- If you use a small roughing tool, then a large amount of material can remain for the finishing tool; keep this in mind when selecting your tools!

Cycle parameters



- **Q215 Machining operation (0/1/2)?**: Define the scope 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 minor 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?**: Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: -360.000 to 360.000. 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





Example

8 CYCL DEF 253 SLOT MILLING	
Q215=0	;MACHINING OPERATION
Q218=80	;SLOT LENGTH
Q219=12	;SLOT WIDTH
Q201=-20	;DEPTH

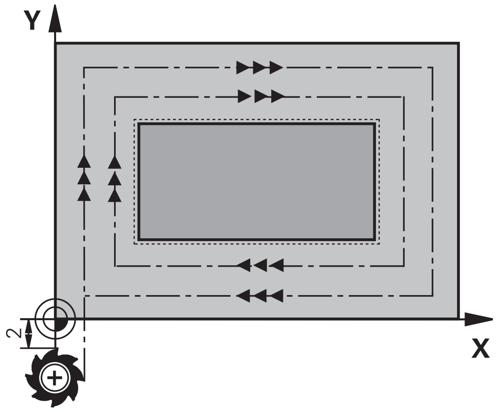
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **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
- **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 control 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

```
Q374=+0     ;SLOT DIRECTION
Q367=0      ;SLOT POSITION
Q202=5      ;PLUNGING DEPTH
Q207=500    ;FEED RATE FOR MILLNG
Q206=150    ;FEED RATE FOR PLNGNG
Q385=500    ;FINISHING FEED RATE
Q338=5      ;INFEED FOR FINISHING
Q200=2      ;SET-UP CLEARANCE
Q203=+0     ;SURFACE COORDINATE
Q204=50     ;2ND SET-UP CLEARANCE
Q351=1      ;CLIMB OR UP-CUT
Q352=0      ;PLUNGE POSITION
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

14.4 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 control 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 control performs a stepover with the current factor, and machines another revolution. The control 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 is retracted from the contour and returns to the starting point of stud machining
6 The control then plunges the tool to the next plunging depth, and machines the stud at this depth
7 This process is repeated until the programmed stud depth is reached.

Please note while programming:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

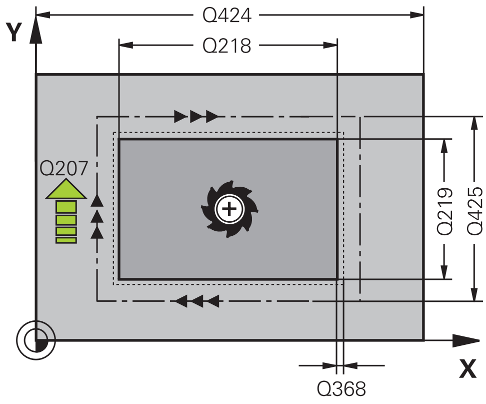
Danger of collision!

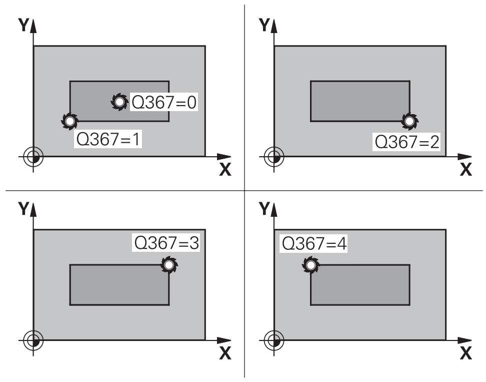
If there is not enough room for the approach movement next to the stud, there is danger of collision.

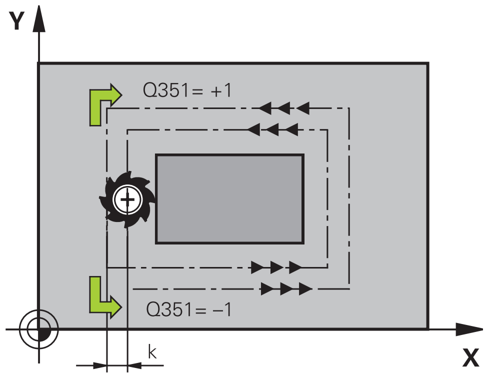
- Depending on the approach position Q439, leave enough room next to the stud for the approach movement
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position.

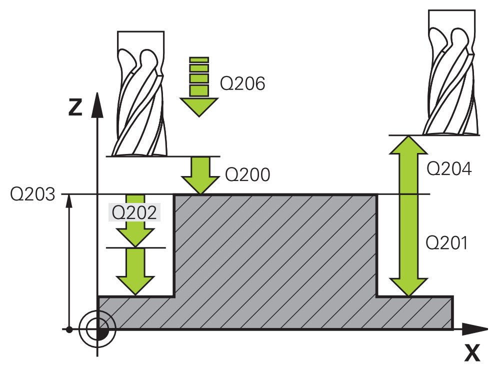
Cycle parameters

- **Q218 First side length?**: Stud length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q424 Workpiece blank side length 1?**: Length of the stud blank, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **First side length**. The control performs multiple lateral 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 control 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 control performs multiple lateral 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 control 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 = lower left corner
  2: Tool position = lower right corner
  3: Tool position = upper right corner
  4: Tool position = upper left corner
- **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 **FMAX**, **FAUTO**, **FU**, **FZ**
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane, is left over after machining. Input range: 0 to 99999.9999









- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999;
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999;
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (if you enter 0, climb milling is performed)
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input range: 0.1 to 1.9999;

Example

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	

14.5 FACE MILLING (Cycle 233)

Cycle run

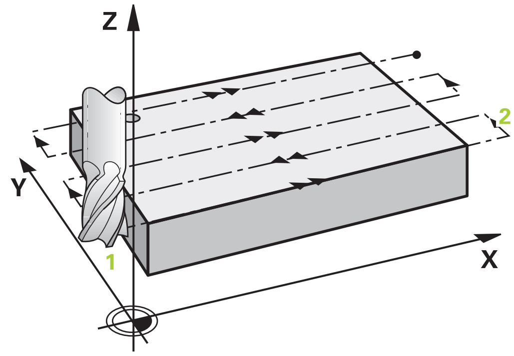
With Cycle 233, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

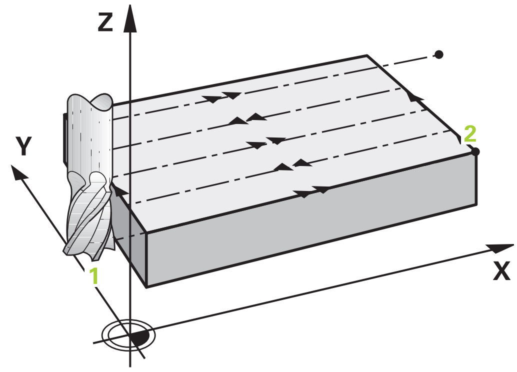
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover after retracting at rapid traverse
- **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 control 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 set-up clearance to the side.
2. The control 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 control

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, with Q389=1, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy Q389=0 is used, the control additionally moves the tool beyond the level surface by the tool radius.

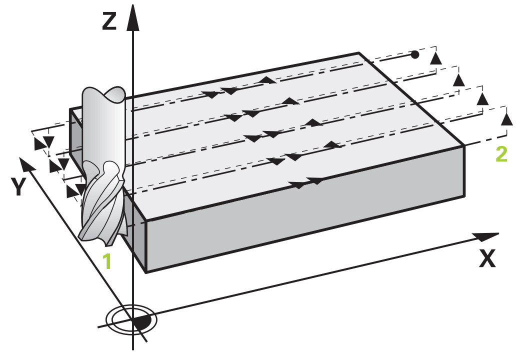


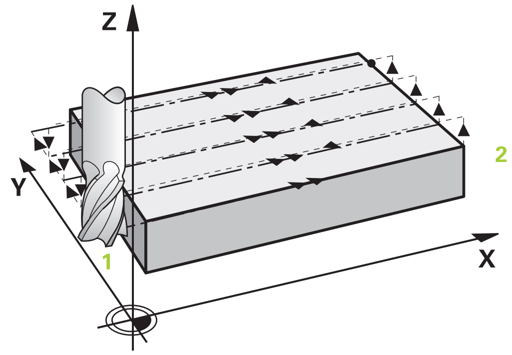


4 The control moves the tool to end point **2** at the programmed feed rate for milling
5 Then the control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side
6 The tool then returns in the opposite direction at the feed rate for milling
7 The process is repeated until the programmed surface has been completed.
8 The control then positions the tool at rapid traverse **FMAX** back to starting point **1**
9 If more than one infeed is required, the control 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 completed. In the last infeed, the programmed finishing allowance will be 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, with Q389=3, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy Q389=2 is used, the control additionally moves the tool beyond the level surface by the tool radius.

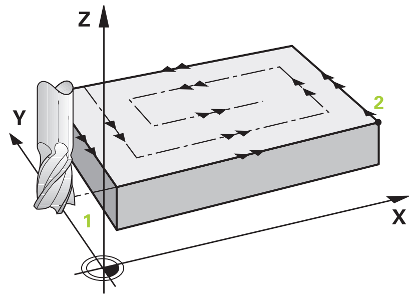




4 The tool subsequently moves to end point **2** at the programmed feed rate for milling
5 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves at **FMAX** paraxially back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side.
6 The tool then returns to the current infeed depth and moves in the direction of end point **2**
7 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**
8 If more than one infeed is required, the control 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 completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate
10 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance

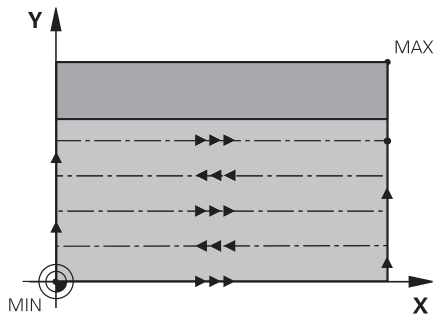
Strategy Q389=4

4 The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a straight line tangential arc
5 The control 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 machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**
7 If more than one infeed is required, the control 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 completed. In the last infeed, the programmed finishing allowance will be 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 limits enable you to set limits to the machining of the level surface so that, for example, side walls or shoulders are considered 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. During roughing, the control takes the allowance for the side into account, whereas during finishing, the allowance is used for pre-positioning 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 control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

Enter **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.

If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

If you define **Q370** TOOL PATH OVERLAP >1, the programmed overlap factor will be taken into account right from the first machining path.

Cycle 233 monitors the entries made for the tool/cutting edge length in **LCUTS** from the tool table. If the tool or cutting edge length is not sufficient for a finishing operation, the control will subdivide the process into multiple machining steps.

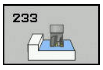
NOTICE

Danger of collision!

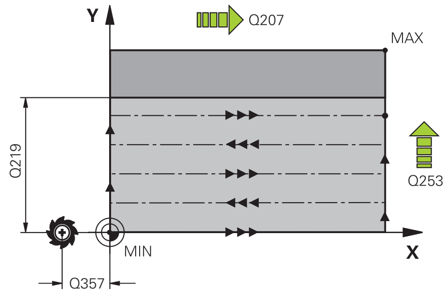
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

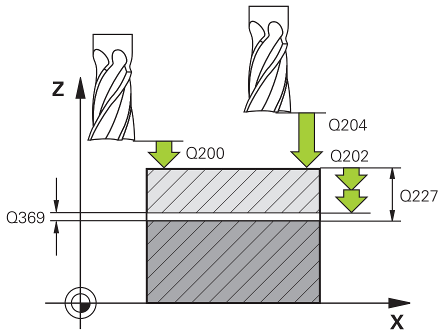
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

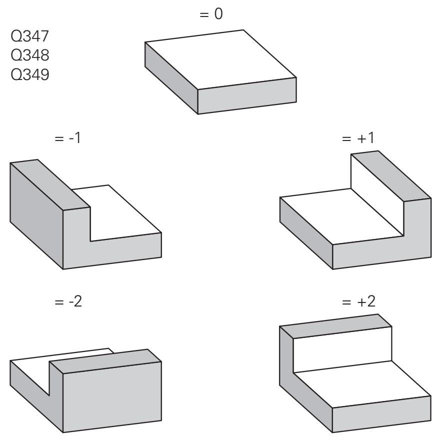
Cycle parameters



- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q389 Machining strategy (0-4)?**: Determine how the control 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 to be machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate outside the surface to be machined
  3: Line-by-line machining, 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 working 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, referencing 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
- **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis to 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 MAX. 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 control calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. Input range: 0.1 to 1.9999.
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min 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 inside the material (Q389=1), the control uses the cross feed rate for milling Q207. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- **Q357 Safety clearance to the side?** (incremental) Parameter Q357 affects the following situations:
  Approaching the first plunging depth: Q357 is the lateral distance between tool and workpiece
  Roughing with the Q389=0-3 roughing strategies: The surface to be machined is extended in **Q350** MILLING DIRECTION by the value from Q357 if no limit has been set in that direction
  Side finishing: The paths will be extended by Q357 in **Q350** MILLING DIRECTION.
  Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999;
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999;

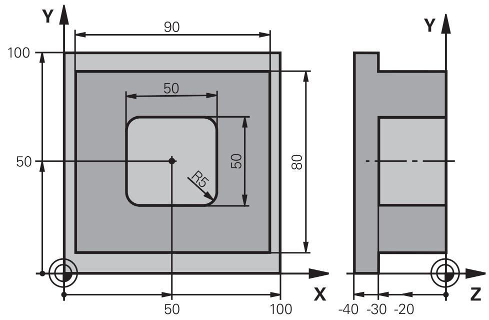
Example

8 CYCL DEF 233 FACE MILLING	
Q215=0	;MACHINING OPERATION
Q389=2	;MILLING STRATEGY
Q350=1	;MILLING DIRECTION
Q218=120	;FIRST SIDE LENGTH
Q219=80	;2ND SIDE LENGTH
Q227=0	;STARTNG PNT 3RD AXIS
Q386=-6	;END POINT 3RD AXIS
Q369=0.2	;ALLOWANCE FOR FLOOR
Q202=3	;MAX. PLUNGING DEPTH
Q370=1	;TOOL PATH OVERLAP
Q207=500	;FEED RATE FOR MILLNG
Q385=500	;FINISHING FEED RATE
Q253=750	;F PRE-POSITIONING
Q357=2	;CLEARANCE TO SIDE
Q200=2	;SET-UP CLEARANCE
Q204=50	;2ND SET-UP CLEARANCE
Q347=0	;1ST LIMIT
Q348=0	;2ND LIMIT
Q349=0	;3RD LIMIT
Q368=0	;ALLOWANCE FOR SIDE
Q338=0	;INFEED FOR FINISHING
Q367=-1	;SURFACE POSITION (-1/0/1/2/3/4)?
9 L X+0 Y+0 R0 FMAX M3 M99	

- **Q347 1st limit?**: Select the side of the workpiece where the level surface is bordered by a side wall. Depending on the position of the side wall, the control limits machining of the level surface to the respective coordinate of the starting point or to the side length: :
  Input **0**: No limiting
  Input **-1**: Limit in negative reference axis
  Input **+1**: Limiting in positive reference axis
  Input **-2**: Limiting in negative minor axis
  Input **+2**: Limiting in positive minor axis
- **Q348 2nd limit?**: See parameter 1st limit Q347
- **Q349 3rd limit?**: See parameter 1st limit Q347
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working 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
- **Q367 Surface position (-1/0/1/2/3/4)?**: Position of the surface referencing the position of the tool when the cycle is called:
  -1: Tool position = current position
  0: Tool position = stud center
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner

14.6 Programming Examples

Example: Milling pockets, studs



0 BEGINN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Call the tool for roughing/finishing
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD	Define cycle for machining the contour outside
Q218=90 ;FIRST SIDE LENGTH	
Q424=100 ;WORKPC. BLANK SIDE 1	
Q219=80 ;2ND SIDE LENGTH	
Q425=100 ;WORKPC. BLANK SIDE 2	
Q201=-30 ;DEPTH	
Q367=0 ;STUD POSITION	
Q202=5 ;PLUNGING DEPTH	
Q207=250 ;FEED RATE FOR MILLNG	
Q206=250 ;FEED RATE FOR PLNGNG	
Q385=750 ;FINISHING FEED RATE	
Q368=0 ;ALLOWANCE FOR SIDE	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q351=+1 ;CLIMB OR UP-CUT	
Q370=1 ;TOOL PATH OVERLAP	
6 X+50 R0	Outside machining
7 Y+50 R0 M3 M99	Call cycle for machining the contour outside
8 CYCL DEF 252 RECTANGULAR POCKET	Define RECTANGULAR POCKET cycle
Q215=0 ;MACHINING OPERATION	
Q218=50 ;FIRST SIDE LENGTH	
Q219=50 ;2ND SIDE LENGTH	

Q201=-30 ;DEPTH	
Q367=+0 ;POCKET POSITION	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE FOR MILLNG	
Q206=150 ;FEED RATE FOR PLNGNG	
Q385=750 ;FINISHING FEED RATE	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q351=+1 ;CLIMB OR UP-CUT	
Q370=1 ;TOOL PATH OVERLAP	
9 X+50 R0 FMAX	
10 Y+50 R0 FMAX M99	Cycle call
11 Z+250 R0 FMAX M30	
12 END PGM C210 MM	

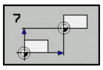
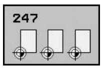
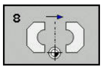
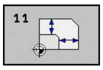
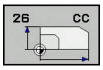
15

Cycles: Coordinate Transformations

15.1 Fundamentals

Overview

Once a contour has been programmed, the control can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The control provides the following functions for coordinate transformations:

Soft key	Cycle	Page
	7 DATUM SHIFT For shifting contours directly within the NC program or from datum tables	391
	247 PRESETTING Presetting during the program run	397
	8 MIRRORING Mirroring contours	398
	11 SCALING FACTOR Increasing or reducing the size of contours	399
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	400

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 NC block (these M functions depend on the machine parameters)
- Select a new NC program

15.2 DATUM SHIFT (Cycle 7)

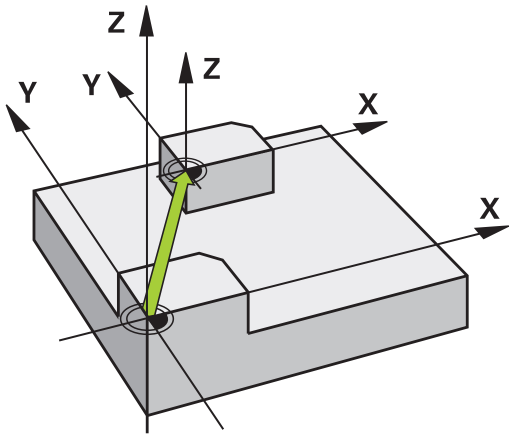
Effect

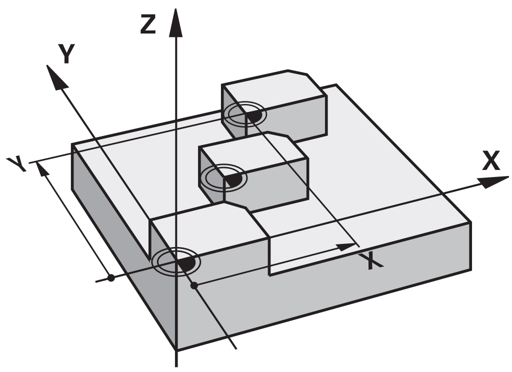
A datum shift allows machining operations to be repeated at various locations on the workpiece.

After the definition of a datum shift cycle, all coordinate data will reference the new datum. The control displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

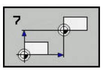
Resetting

- To shift the datum back to the coordinates X=0, Y=0 etc., program another 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 reference the workpiece datum defined by presetting. Incremental values always reference 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

Example

```
13 CYCL DEF 7.0  DATUM SHIFT
14 CYCL DEF 7.1  X+60
15 CYCL DEF 7.2  Y+40
16 CYCL DEF 7.3  Z-5
```

Please note while programming

Refer to your machine manual!

Possible datum shift values in the rotary axes will be specified by your machine tool builder in the **presetToAlignAxis** parameter (no. 300203).

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

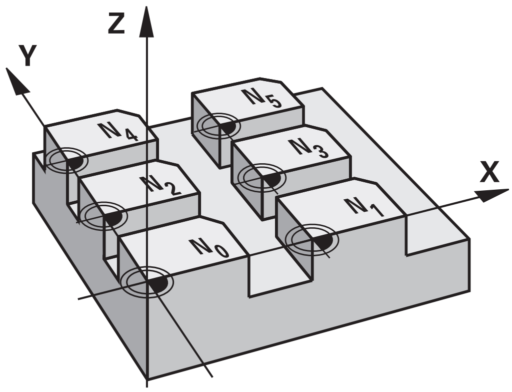
15.3 DATUM SHIFT with datum tables (Cycle 7)

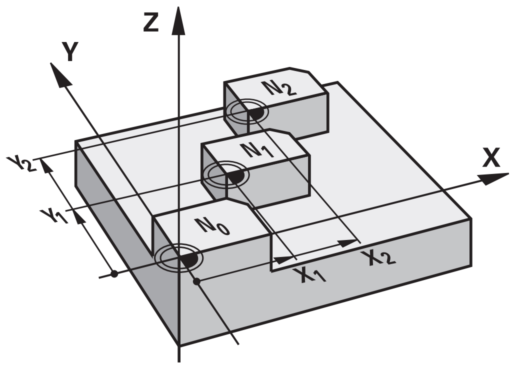
Effect

Datum tables are used for:

- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within an NC 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.
- To shift the datum back to the coordinates X=0, Y=0 etc., directly call a cycle definition

Status displays

In the additional status display, the following data from the datum table is shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

Please note while programming:

Datums from a datum table **always and exclusively** reference the current preset.

If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

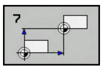
If you work without **SEL TABLE,** then you must activate the desired datum table before the test run or the program run (this applies also to the program run):

- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table now has the status S
- Use the file management to select the desired table for the **Program run, single block** and **Program run, full sequence** operating modes: The table now has 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 in a Q parameter. If you enter a Q parameter, the control activates the datum number entered in the Q parameter. Input range: 0 to 9999

Example

```
77 CYCL DEF 7.0  DATUM SHIFT
78 CYCL DEF 7.1  #5
```

Selecting a datum table in the part program

With the **SEL TABLE** function, you select the datum table from which the control 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
- Enter the complete path name of the datum table or select the file with the **SELECT** soft key. Confirm your input 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 will not be taken into account when the NC program is executed.

Select the datum table in the **Programming** mode of operation

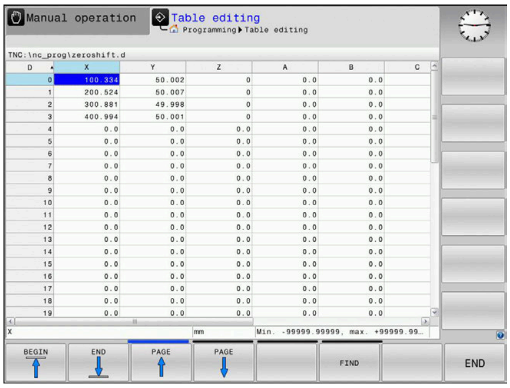
PGM MGT

- 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 in the soft-key row include:

Soft key	Function
BEGIN	Select the beginning of the table
END	Select the table end
PAGE	Go to previous page
PAGE	Go to next page
INSERT LINE	Insert line
DELETE LINE	Delete line
FIND	Find
BEGIN LINE	Move the cursor to the beginning of the line
END LINE	Move the cursor to the end of the line
COPY FIELD	Copy the current value
PASTE FIELD	Insert the copied value
APPEND N LINES AT END	Add the entered number of lines (datums) to the end of the table

Configuring a datum table

If you do not wish to define a datum for an active axis, press the **CE** key. Then the control clears the numerical value from the corresponding input field.



> You can change the properties of tables. Enter code number 555343 in the MOD menu. The control then displays the **EDIT FORMAT** soft key if a table is selected. When you press this soft key, the control opens a pop-up window where the properties are shown for each column of the selected table. Any changes you make only affect the open table.

Leaving a datum table

Select a different type of file in file management. Select the desired file.

NOTICE

Danger of collision!

The control considers changes in a datum table only when the values are saved.

- Make sure to confirm any changes made to the table immediately by pressing the **ENT** key
- Carefully test the NC program after making a change to the datum table

Status displays

In the additional status display, the control shows the values of the active datum shift.

15.4 PRESETTING (Cycle 247)

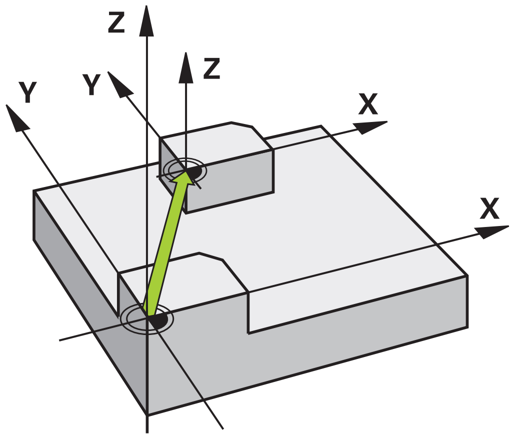
Effect

With the presetting cycle you can activate as the new preset a preset defined in the preset table.

After a presetting cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

Status display

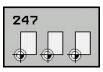
In the status display; the control shows the active preset number behind the preset symbol.



Please note before programming:

> When activating a preset from the preset table, the control resets the datum shift, mirroring, scaling factor and axis-specific scaling factor.
>
> If you activate preset number 0 (line 0), then you activate the preset that you last set in the **Manual operation** or **Electronic handwheel** operating mode.
>
> Cycle 247 is also effective in the Test Run operating mode.

Cycle parameters



- **Number for preset?**: Enter the number of the desired preset from the preset table. Alternatively, you can press the **SELECT** soft key and directly select the desired preset from the preset table. Input range: 0 to 65535

Example

```
13 CYCL DEF 247  PRESETTING
    Q339=4       ;PRESET NUMBER
```

15.5 MIRRORING (Cycle 8)

Effect

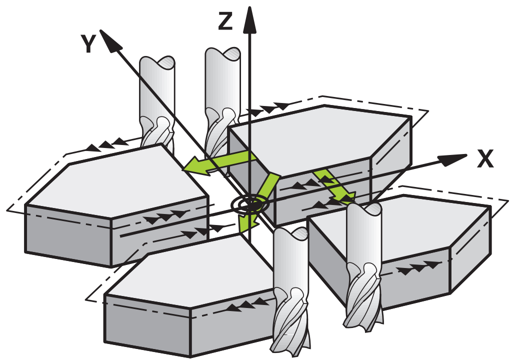
The control can machine the mirror image of a contour in the working plane.

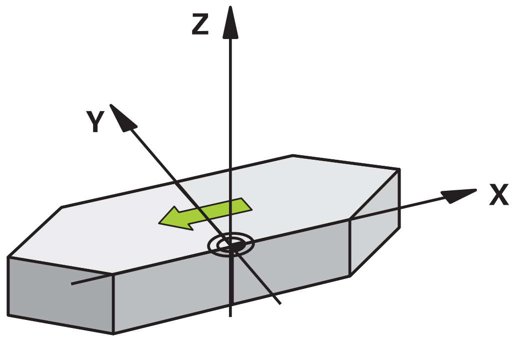
Mirroring becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active mirrored axes are shown in the additional status display.

- 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.

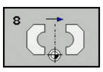




Resetting

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—including rotary axes —except for the spindle axis and its associated secondary axis. You can enter up to three axes. Input range: up to three NC axes **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example

```
79 CYCL DEF 8.0  MIRRORING
80 CYCL DEF 8.1  X Y  Z
```

15.6 SCALING (Cycle 11)

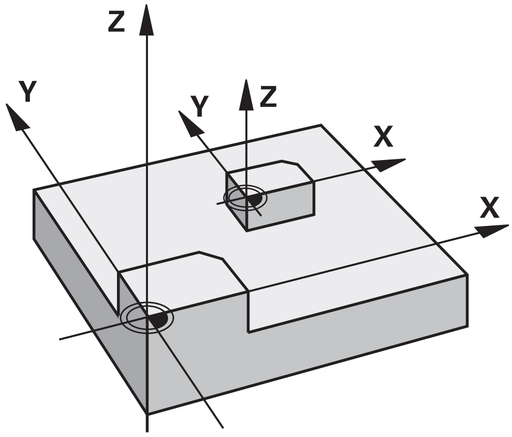
Effect

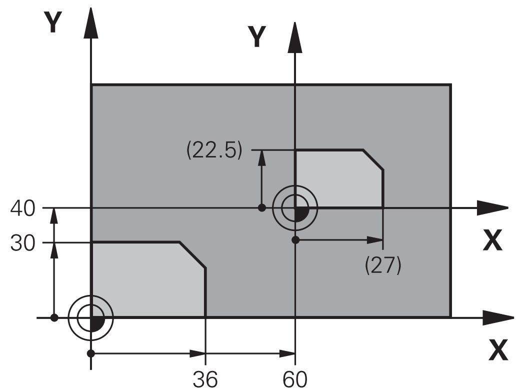
The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The factor defined for SCALING becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. 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.

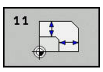
Enlargement: SCL greater than 1 (up to 99.999 999)

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 control multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range: 0.000001 to 99.999999

Example

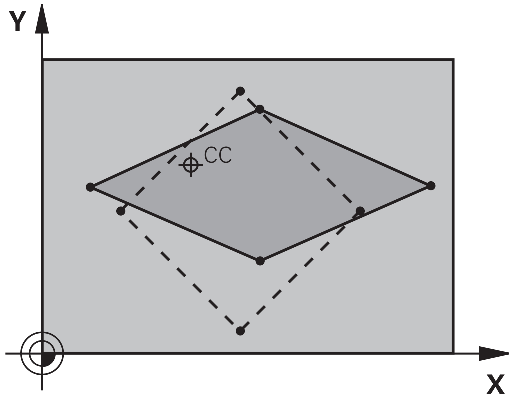
11 CALL LBL 1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1

15.7 AXIS-SPECIFIC SCALING (Cycle 26)

Effect

With Cycle 26, you can account for shrinkage and oversize factors for each axis.

The factor defined for SCALING becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. 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 corresponding axis.

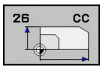
Please note while programming:

You can program each coordinate axis with its own axis-specific scaling factor.

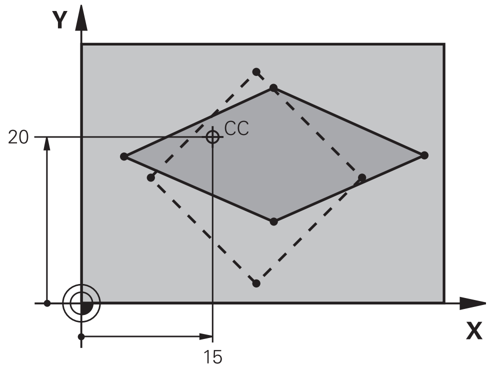
In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.

Cycle parameters



- **Axis and factor**: Select the coordinate axis/axes via soft key. Enter the factor(s) for axis-specific enlargement or reduction. 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



Example

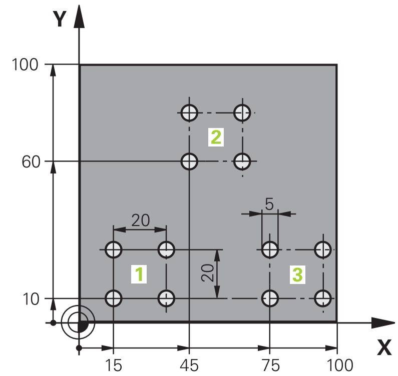
25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL 1

15.8 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	

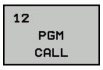
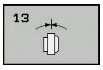
16

Cycles: Special Functions

16.1 Fundamentals

Overview

The control provides the following cycles for the following special purposes:

Soft key	Cycle	Page
	9 DWELL TIME	407
	12 Program call	408
	13 Oriented spindle stop	409

16.2 DWELL TIME (Cycle 9)

Function

Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.

The cycle becomes effective as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Example

```
89 CYCL DEF 9.0  DWELL TIME
90 CYCL DEF 9.1  DWELL 1.5
```

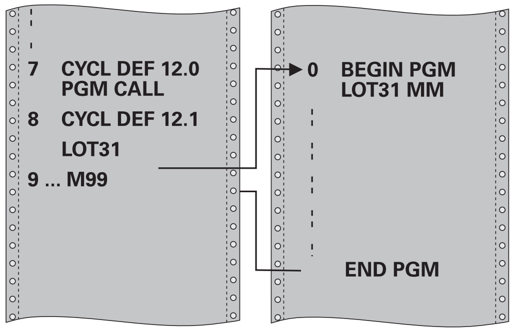
Cycle parameters

- **Dwell time in seconds**: Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

16.3 PROGRAM CALL (Cycle 12)

Cycle function

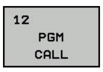
NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.



Please note while programming:

> The NC program you are calling must be stored in the internal memory of your control.
>
> If the NC program you are defining to be a cycle is located in the same directory as the NC program you are calling it from, you need only enter the program name.
>
> If the NC program you are defining to be a cycle is not located in the same directory as the NC 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 NC program can also influence the calling NC program.

Cycle parameters



- **Program name**: Enter the name of the NC program and, if necessary, the path where it is located, or
- Activate the file select dialog with the **SELECT** soft key. Select the NC program to be called.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare program 50.i 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

16.4 SPINDLE ORIENTATION (Cycle 13)

Cycle function

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

The control can control the main machine tool spindle and rotate it to a given angular position.

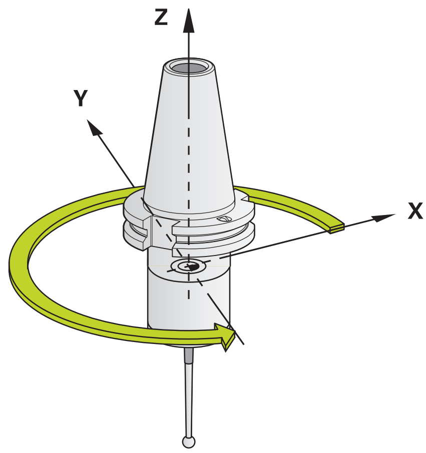
Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

With M19 or M20, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program M19 or M20 without having defined Cycle 13 beforehand, the control positions the main spindle at an angle that has been set by the machine tool builder.

More information: machine tool manual.



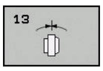
Example

```
93 CYCL DEF 13.0  ORIENTATION
94 CYCL DEF 13.1  ANGLE 180
```

Please note while programming:

Cycle 13 is used internally for Cycles 202 and 204. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

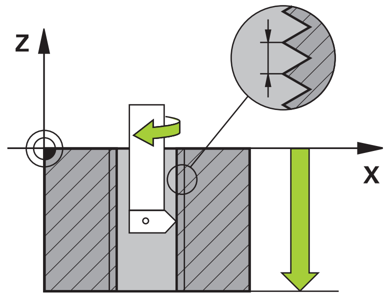


- **Angle of orientation**: Enter the angle with reference to the angle reference axis of the working plane. Input range: 0.0000° to 360.0000°

16.5 THREAD CUTTING (Cycle 18)

Cycle run

Cycle **18** THREAD CUTTING moves the tool with servo-controlled spindle from the momentary position with active speed to the specified depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.



Please note while programming:

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (speed override is not active). The Control then adapts the spindle speed as required.
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
- **limitSpindleSpeed** (no. 113604): Spindle speed limit
  True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time
  False: (Limiting not active)

The spindle speed potentiometer is inactive.

Before calling this cycle, be sure to program a spindle stop! (For example with M5.) The control automatically activates spindle rotation at the start of the cycle and deactivates it at the end.

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

NOTICE

Danger of collision!

A collision may occur if you do not program pre-positioning before calling Cycle 18. Cycle 18 does not perform approach and departure motion.

- Pre-position the tool before the start of the cycle.
- The tool moves from the current position to the entered depth after the cycle is called

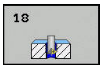
NOTICE

Danger of collision!

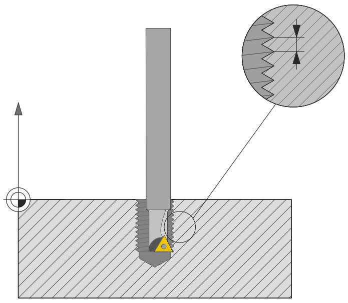
If the spindle was switched on before calling the cycle, Cycle 18 switches the spindle off and the cycle works with a stationary spindle! Cycle 18 switches the spindle on again at the end if it was switched on before cycle start.

- Before starting this cycle, be sure to program a spindle stop! (For example with M5.)
- At the end of Cycle 18, the control restores the spindle to its state at cycle start. If the spindle was switched off before this cycle, the control will switch it off again at the end of Cycle 18.

Cycle parameters



- Boring depth (incremental): Enter the thread depth based on the current position. Input range: -99999 ... +99999
- Thread pitch: Enter the pitch of the thread. The algebraic sign entered here differentiates between right-hand and left-hand threads:
  + = right-hand thread (M3 with negative hole depth)
  - = left-hand thread (M4 with negative hole depth)



Example

25 CYCL DEF 18.0 THREAD CUTTING
26 CYCL DEF 18.1 DEPTH = -20
27 CYCL DEF 18.2 PITCH = +1

17

Touch probe cycles

17.1 General information about touch probe cycles

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

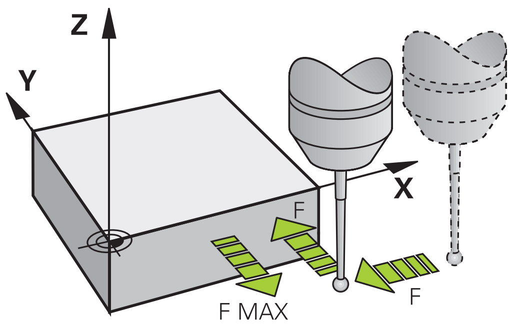
The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

The touch probe cycles are available only with option 17. If you are using a HEIDENHAIN touch probe, this option is automatically available.

Method of function

Whenever the control 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 415



When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the control: 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 control 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 control provides touch probe cycles that allow you to:

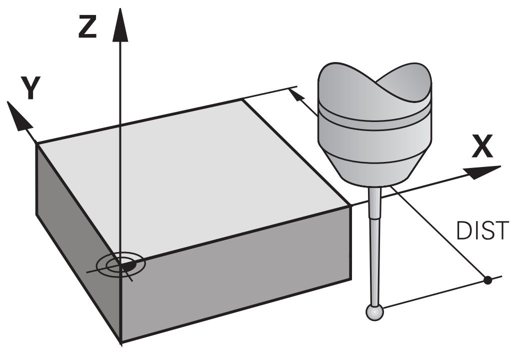
- Calibrate the touch probe
- Set presets

17.2 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.

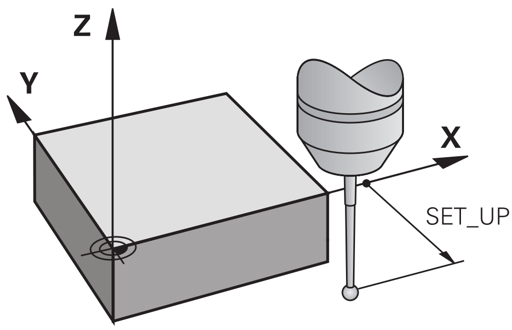
Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the range defined in **DISST**, the control will issue 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 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 **SET_UP**.



Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table

To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

> If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch trigger probe, probing feed rate: F in touch probe table

In **F**, you define the feed rate at which the control is to probe the workpiece.

F can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).

The feed rate potentiometer may be effective with touch probe cycles. The machine tool builder defines the required settings. (the parameter **overrideForMeasure** (No. 122604) must be appropriately configured.)

Touch trigger probe, rapid traverse for positioning: FMAX

In **FMAX**, you define the feed rate at which the control pre-positions the touch probe and 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 control 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 control runs the cycle automatically as soon as it executes the cycle definition in the program run.

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 1400 to 1499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **8 MIRROR IMAGE**, Cycles **11 SCALING** and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand.

Touch probe cycles numbered 400 to 499 or 1400 to 1499 position the touch probe according to the following positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to clearance height and then positions it in the working plane to the first touch point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the control 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.

17.3 Touch-probe table

General information

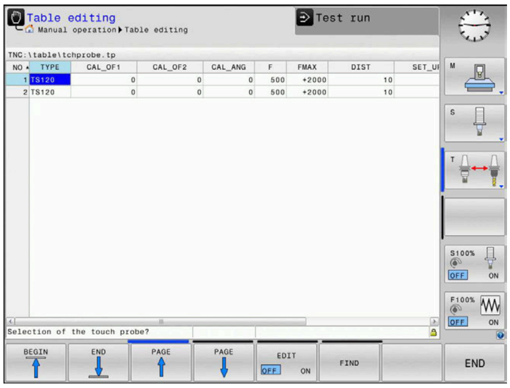
Various data is stored in the touch probe table that defines the probe behavior during the probing process. If you use several touch probes on your machine tool, you can save separate data for each touch probe.

> You can also view and edit the data of the touch probe table in the expanded tool management (option 93).

Editing touch probe tables

To edit the touch probe table, proceed as follows:

- Operating mode: Press the **Manual operation** key
- Select the touch probe functions: Press the **TOUCH PROBE** soft key. The control displays additional soft keys.
- Select the touch probe table: Press the **TCH PROBE TABLE** soft key.
- Set the **EDIT** soft key to **ON**.
- Using the arrow keys, select the desired setting.
- Perform desired changes.
- Exit the touch probe table: Press the **END** soft key.



Touch probe data

Abbr.	Inputs	Dialog
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool number	–
TYPE	Selection of the touch probe used	**Selection of the touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis in the principal axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis in the minor axis	**TS center misalignmt. aux. axis? [mm]**
CAL_ANG	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the control will probe the workpiece **F** can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe is pre-positioning and is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within this defined value, the control will issue an error message.	**Maximum measuring range? [mm]**
SET_UP	In **set_up** you define how far from the defined or calculated touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to the **SET_UP** machine parameter.	**Set-up clearance? [mm]**
F_PREPOS	Defining speed with pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-position at rapid? ENT/NOENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probing process. In this way, the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Probe oriented? Yes=ENT/No=NOENT**
SERIAL	You need not make an entry in this column. The TNC automatically enters the serial number of the touch probe if the touch probe has an EnDat interface.	**Serial number?**
REACTION	Behavior in case of collision with the touch probe ■ **NCSTOP**: The NC program will be aborted. ■ **EMERGSTOP**: Emergency stop, safe braking of the axes.	**Reaction?**

17.4 Fundamentals

Overview

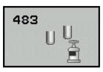
> **i** Operating notes
> - When running touch probe cycles, Cycle **8 MIRROR IMAGE**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active.
> - HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.

> The control and the machine tool must be set up by the machine tool builder for use of the TT touch probe.
>
> Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.
>
> The touch probe cycles are available only with the Touch Probe Functions software option (option 17).

In conjunction with the control'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.T tool file and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Measurement of a stationary tool
- Measurement of a rotating tool
- Measurement of individual teeth

You can program the cycles for tool measurement in **Programming** mode of operation using the **CYCL DEF** key. The following cycles are available:

Soft key	Cycle	Page
	Calibrating the TT, Cycle 480	424
	Measuring the tool length, Cycle 481	428
	Measuring the tool radius, Cycle 482	430
	Measuring the tool length and radius, Cycle 483	432

> **i** The measuring cycles can be used only when the central tool file TOOL.T is active.
>
> Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.

Setting machine parameters

> Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200).
>
> Touch probe cycles 480, 481, 482, 483 and 484 can be hidden with the machine parameter **hideMeasureTT** (No. 128901).
>
> When measuring a stationary tool, the Control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius [mm]

The probing feed rate is calculated as follows:
v = measuring tolerance • n with

v:	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate:

probingFeedCalc (No. 122710) = **ConstantTolerance**:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance (**measureTolerance1** (no. 122715), the sooner you will encounter this effect.

probingFeedCalc (No. 122710) = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc (No. 122710) = **ConstantFeed**:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance = r • **measureTolerance1**/ 5 mm, where

r:	Active tool radius [mm]
measureTolerance1:	Maximum permissible error of measurement

Entries in the tool table TOOL.T

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT.	Cutting direction of the tool for measuring a rotating tool	**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	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to **offsetToolAxis**. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill	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** (no. 122707) used)
Radius cutter e.g. with a diameter of 10 mm	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured	5 (always define an offset at least equal to the tool radius in order to make sure that the measured diameter is correct)

17.5 Calibrating the TT (Cycle 480, option 17)

Cycle run

The TT is calibrated with measuring cycle TCH PROBE 480. . The calibration process runs automatically. The control 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 cylindrical pin. The resulting calibration values are stored in the control memory and are accounted for during subsequent tool measurement.

Calibration process:

1. Clamp the calibrating tool. The calibrating tool must be a precisely cylindrical part, for example a cylinder pin
2. Manually position the calibrating tool in the working plane via the center of the TT
3. Position the calibrating tool in the tool axis approx. 15 mm + safety clearance above the TT
4. The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
5. The calibration process along the tool axis starts
6. Calibration then follows in the working plane
7. The control positions the calibrating tool in the working plane at a position of TT radius + set-up clearance + 11 mm
8. Then the TNC moves the tool downwards along the tool axis and the calibration process starts
9. During probing, the control moves in a square pattern
10. The control saves the calibration values and considers them during subsequent tool measurement
11. The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Please note while programming:

The functioning of the calibration cycle is dependent on machine parameter **CfgTTRoundStylus** (No. 114200). Refer to your machine manual.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the TOOL.T tool table.

The position of the TT within the machine working space must be defined by setting machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**.

If you change the setting of any of the machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**, you must recalibrate.

Cycle parameters

- **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistToolAx** (no. 114203)). Input range: -99999.9999 to 99999.9999

Example of new format

```
6 TOOL CALL  1 Z
7 TCH PROBE 480  CALIBRATE TT
    Q260=+100 ;CLEARANCE HEIGHT
```

17.6 Calibrating the wireless TT 449 (Cycle 484, Option 17)

Fundamentals

With Cycle 484, you can calibrate your tool touch probe, e.g the wireless infrared TT 449 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle 484 you must move the tool over the TT

Cycle run

To calibrate the tool touch probe, program measuring cycle TCH PROBE 484. In 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 control interrupts the calibration cycle.
- The control displays a dialog in a new window.
- You are prompted 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 cylindrical pin. Enter the exact length and radius of the calibrating tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Please note while programming:

NOTICE

Danger of collision!

To avoid collisions the tool must be pre-positioned before calling the cycle with **Q536**=1! The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

- Specify whether to stop before cycle start or run the cycle automatically without stopping.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. When using a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. The use of a calibrating tool of too small a diameter and/or protruding too far from the chuck may cause significant inaccuracies.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

- **Q536 Stop before running (0=Stop)?**: Specify whether to stop before cycle start or run the cycle automatically without stopping:
  0: Stop before running the cycle. You are prompted in a dialog 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 control starts the calibration process from the current position. Before running Cycle 484, you must position the tool above the tool touch probe.

17.7 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 stationary.
- 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 stationary.

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 (**R-OFFS**).

Cycle for measuring a stationary tool (e.g. for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

Cycle for measuring individual teeth

The control 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** (no. 122707). You can enter an additional offset in Tool offset: Length (**L-OFFS**) in the tool table. The control probes the tool radially while it is rotating to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation.

Please note while programming:

> Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: 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 tool table TOOL.T, and the tool compensation is set to DL=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool length is compared to the tool length L from TOOL.T. The control 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 control 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 control calculates the deviation from the stored value and writes it to 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 references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top 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).

Example

6 TOOL CALL 12 Z
7 TCH PROBE 481 CAL. TOOL LENGTH
Q340=1 ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH

17.8 Measuring a tool radius (Cycle 482, option 17)

Cycle run

To measure a tool radius, program the measuring cycle TCH PROBE 482. Select via input parameters by which of two methods the tool radius 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 control 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 control probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control will measure 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.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, define the number of teeth **CUT** as 0 in the tool table and adjust machine **CfgTT** (no. 122700). Refer to your machine manual.

Cycle parameters

- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool radius is compared to the tool radius R from TOOL.T. The control 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 control 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 control calculates the deviation from the stored value and writes it to 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 references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top 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).

Example

6 TOOL CALL 12 Z
7 TCH PROBE 482 CAL. TOOL RADIUS
Q340=1 ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH

17.9 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 control 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 481 and 482.

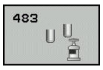
Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgTT** (No. 122700) Refer to your machine manual.

Cycle parameters



- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL=0 and DR=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control 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 control 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 control calculates the deviation from the stored values and writes it to Q parameter Q115 or 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 references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top 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).

Example

6 TOOL CALL 12 Z
7 TCH PROBE 483 MEASURE TOOL
Q340=1 ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1 ;PROBING THE TEETH

18

Tables and Overviews

18.1 System data

List of FN 18 functions

With the **FN 18: SYSREAD** function, you can read system data and save them to Q parameters. The selection of the system datum occurs via a group number (ID no.), a system data number, and, if necessary, an index.

> The read values of the function **FN 18: SYSREAD** are always output by the control in **metric** units regardless of the NC program's unit of measure.

The following is a complete list of the **FN 18: SYSREAD** function. Please be aware that not all functions are available depending on the model of your control.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Program information				
	10	3	-	Number of the active machining cycle
		6	-	Number of the most recently executed touch probe cycle –1 = None
		7	-	Type of calling NC program: –1 = None 0 = Visible NC program 1 = Cycle/macro, main program is visible 2 = Cycle/macro, there is no visible main program
		103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
		110	QS parameter number	Is there a file with the name QS(IDX)? 0 = No, 1 = Yes This function eliminates relative file paths.
		111	QS parameter number	Is there a directory with the name QS(IDX)? 0 = no, 1 = Yes Only absolute directory paths are possible.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
System jump addresses				
	13	1	-	Label number or label name (string or QS) jumped to during M2/M30 instead of ending the current NC program. Value = 0: M2/M30 have the normal effect
		2	-	Label number or label name (string or QS) jumped to in the event of FN14: ERROR with the NC CANCEL reaction instead of aborting the NC 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 number or label name (string or QS) jumped to in the event of an internal server error (SQL, PLC, CFG) or with erroneous file operations (FUNCTION FILECOPY, FUNCTION FILEMOVE, or FUNCTION FILEDELETE) instead of aborting the NC program with an error message. Value = 0: Error has the normal effect.
Machine status				
	20	1	-	Active tool number
		2	-	Prepared tool number
		3	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W
		4	-	Programmed spindle speed
		5	-	Active spindle condition -1 = spindle condition not defined 0 = M3 active 1 = M4 active 2 = M5 active after M3 3 = M5 active after M4
		7	-	Active gear range
		8	-	Active coolant status 0 = off, 1 = on
		9	-	Active feed rate
		10	-	Index of prepared tool
		11	-	Index of active tool
		14	-	Number of active spindle
		20	-	Programmed cutting speed in turning operation
		21	-	Spindle mode in turning mode: 0 = constant speed 1 = constant cutting speed

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		22	-	Coolant status M7: 0 = inactive, 1 = active
		23	-	Coolant status M8: 0 = inactive, 1 = active
Channel data				
	25	1	-	Channel number
Cycle parameters				
	30	1	-	Set-up clearance
		2	-	Hole depth / milling depth
		3	-	Plunging depth
		4	-	Feed rate for plunging
		5	-	First side length of pocket
		6	-	Second side length of pocket
		7	-	First side length of slot
		8	-	Second side length of slot
		9	-	Radius of circular pocket
		10	-	Feed rate for milling
		11	-	Rotational direction of the milling path
		12	-	Dwell time
		13	-	Thread pitch for Cycles 17 and 18
		14	-	Finishing allowance
		15	-	Roughing angle
		21	-	Probing angle
		22	-	Probing path
		23	-	Probing feed rate
		49	-	HSC mode (Cycle 32 Tolerance)
		50	-	Tolerance for rotary axes (Cycle 32 Tolerance)
		52	Q parameter number	Type of transfer parameter for user cycles: –1: Cycle parameter not programmed in CYCL DEF 0: Cycle parameter numerically programmed in CYCL DEF (Q parameter) 1: Cycle parameter programmed as string in CYCL DEF (Q parameter)
		60	-	Clearance height (touch probe cycles 30 to 33)
		61	-	Inspection (touch probe cycles 30 to 33)
		62	-	Cutting edge measurement (touch probe cycles 30 to 33)
		63	-	Q parameter number for the result (touch probe cycles 30 to 33)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		64	-	Q parameter type for the result (touch probe cycles 30 to 33) 1 = Q, 2 = QL, 3 = QR
		70	-	Multiplier for feed rate (cycles 17 and 18)
Modal status				
	35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
Data for SQL tables				
	40	1	-	Result code for the last SQL command. If the last result code was 1 (=error), the error code is transferred as the return code.
Data from the tool table				
	50	1	Tool no.	Tool length L
		2	Tool no.	Tool radius R
		3	Tool no.	Tool radius R2
		4	Tool no.	Oversize for tool length DL
		5	Tool no.	Tool radius oversize DR
		6	Tool no.	Tool radius oversize DR2
		7	Tool no.	Tool locked TL 0 = not locked, 1 = locked
		8	Tool no.	Number of the replacement tool RT
		9	Tool no.	Maximum tool age TIME1
		10	Tool no.	Maximum tool age TIME2
		11	Tool no.	Current tool age CUR.TIME
		12	Tool no.	PLC status
		13	Tool no.	Maximum tooth length LCUTS
		14	Tool no.	Maximum plunge angle ANGLE
		15	Tool no.	TT: Number of tool teeth CUT
		16	Tool no.	TT: Wear tolerance for length, LTOL
		17	Tool no.	TT: Wear tolerance for radius, RTOL
		18	Tool no.	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	Tool no.	TT: Offset in plane R-OFFS R = 99999.9999
		20	Tool no.	TT: Offset in length L-OFFS
		21	Tool no.	TT: Breakage tolerance for length, LBREAK
		22	Tool no.	TT: Breakage tolerance for radius, RBREAK
		28	Tool no.	Maximum speed NMAX
		32	Tool no.	Point angle TANGLE

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		34	Tool no.	LIFTOFF allowed (0 = No, 1 = Yes)
		35	Tool no.	Wear tolerance for radius R2TOL
		36	Tool no.	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	Tool no.	Corresponding line in the touch-probe table
		38	Tool no.	Timestamp of last use
		40	Tool no.	Pitch for thread cycles

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Data from the pocket table				
	51	1	Pocket number	Tool number
		2	Pocket number	0 = no special tool 1 = special tool
		3	Pocket number	0 = no fixed pocket 1 = fixed pocket
		4	Pocket number	0 = pocket not locked 1 = pocket locked
		5	Pocket number	PLC status
Determine the tool pocket				
	52	1	Tool no.	Pocket number
		2	Tool no.	Tool magazine number
Tool data for T and S strobes				
	57	1	T code	Tool number IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		2	T code	Tool index IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		5	-	Spindle speed IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
Values programmed in TOOL CALL				
	60	1	-	Tool number T
		2	-	Active tool axis 0 = X 1 = Y 2 = Z 6 = U 7 = V 8 = W
		3	-	Spindle speed S
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Automatic TOOL CALL 0 = Yes, 1 = No
		7	-	Tool radius oversize DR2
		8	-	Tool index
		9	-	Active feed rate
		10	-	Cutting speed [mm/min]

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values for LAC and VSC				
	71	0	2	Total inertia determined by the LAC weighing run in [kgm²] (with A/B/C rotary axes) or total mass in [kg] (with X/Y/Z linear axes)
		1	0	Cycle 957 Retraction from thread
Freely available memory area for OEM cycles				
	72	0-39	0 to 30	Freely available memory area for OEM cycles. The values are only reset by the control during a control reboot (= 0). With "Cancel," the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Freely available memory area for user cycles				
	73	0-39	0 to 30	Freely available memory area for user cycles The values are only reset by the control during a control reboot (= 0). With "Cancel," the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Read minimum and maximum spindle speed				
	90	1	Spindle ID	Minimum spindle speed of the lowest gear range. If no gear stages are configured, CfgFeedLimits/minFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle
		2	Spindle ID	Maximum spindle speed from the highest gear stage. If no gear stages are configured, CfgFeedLimits/maxFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle
Tool compensation				
	200	1	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active radius
		2	1 = without oversize 2 = with oversize 3 = with	Active length

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			oversize and oversize from TOOL CALL	
		3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
		6	Tool no.	Tool length Index 0= active tool
Coordinate transformations				
	210	1	-	Basic rotation (manual)
		2	-	Programmed rotation
		3	-	Active mirror axis. Bits 0 to 2 and 6 to 8: Axes X, Y, Z and U, V, W
		4	Axis	Active scaling factor Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 - 3 (A, B, C)
		6	-	Tilt working plane in Program Run operating modes 0 = Not active –1 = Active
		7	-	Tilt working plane in Manual operating modes 0 = Not active –1 = Active
		8	QL parameter no.	Angle of misalignment between spindle and tilted coordinate system. Projects the angle specified in the QL parameter from the input coordinate system to the tool coordinate system. If IDX is omitted, the angle 0 is used for projection.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Active coordinate system				
	211	–	-	1 = input system (default) 2 = REF system 3 = tool change system
Special transformations in turning mode				
	215	1	-	Angle for the precession of the input system in the XY plane in turning mode To reset the transformation the value 0 must be entered for the angle. This transformation is used in connection with Cycle 800 (parameter Q497).
		3	1-3	Reading out of the spatial angle written with NR2 Index: 1 - 3 (redA, redB, redC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read values for OEM offset.. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	-	Software limit switch on or off: 0 = on, 1 = off For modulo axes, either both the upper and lower limits or no limit at all must be set.
Read the nominal position in the REF system				
	240	1	Axis	Current nominal position in the REF system
Read the nominal position in the REF system, including offsets (handwheel, etc.)				
	241	1	Axis	Current nominal position in the REF system
Read the current position in the active coordinate system				
	270	1	Axis	Current nominal position in the input system When called while tool radius compensation is active, the function supplies the uncompensated positions for the principal axes X, Y, and Z. If the function is called for a rotary axis and tool radius compensation is active, an error message is issued. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
Read the current position in the active coordinate system, including offsets (handwheel, etc.)				
	271	1	Axis	Current nominal position in the input system

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read information to M128				
	280	1	-	M128 active: –1 = Yes, 0 = No
		3	-	Condition of TCPM after Q No.: Q No. + 0: TCPM active, 0 = no, 1 = yes Q No. + 1: AXIS, 0 = POS, 1 = SPAT Q No. + 2: PATHCTRL, 0 = AXIS, 1 = VECTOR Q No. + 3: Feed rate, 0 = F TCP, 1 = F CONT
Machine kinematics				
	290	5	-	0: Temperature compensation not active 1: Temperature compensation active
		10	-	Index of the machine kinematics from Channels/ChannelSettings/CfgKin-List/kinCompositeModels programmed in FUNCTION MODE MILL or FUNCTION MODE TURN –1 = Not programmed.
Read data of the machine kinematics				
	295	1	QS parameter no.	Read the axis names of the active 3-axis kinematics. The axis names are written according to QS(IDX), QS(IDX+1), and QS(IDX+2). 0 = Operation successful
		2	0	Is FACING HEAD POS function active? 1 = Yes, 0 = No
		4	Rotary axis	Read whether the defined rotary axis participates in the kinematic calculation. 1 = Yes, 0 = No (A rotary axis can be excluded from the kinematics calculating using M138.) Index: 4, 5, 6 (A, B, C)
		6	Axis	Angle head: Displacement vector in the basic coordinate system B-CS through angle head Index: 1, 2, 3 (X, Y, Z)
		7	Axis	Angle head: Direction vector of the tool in the basic coordinate system B-CS Index: 1, 2, 3 (X, Y, Z)
		10	Axis	Determine programmable axes. Determine the axis ID associated with the specified axis index (index from CfgAxis/axisList). Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		11	Axis ID	Determine programmable axes. Determine the index of the axis (X = 1, Y = 2, ...) for the specified axis ID Index: Axis ID (index from CfgAxis/axisList)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modify the geometrical behavior				
	310	20	Axis	Diameter programming: –1 = on, 0 = off
Current system time				
	320	1	0	System time in seconds that has elapsed since 01.01.1970, 00:00:00 (real time).
			1	System time in seconds that has elapsed since 01.01.1970, 00:00:00 (look-ahead calculation).
		3	-	Read the processing time of the current NC program.
Formatting of system time				
	321	0	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm:ss
		1	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm:ss
		2	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm
		3	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY h:mm
		4	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm:ss

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm:ss
		5	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm
		6	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD h:mm
		7	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD h:mm
		8	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY
		9	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY
		10	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY
		11	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD
		12	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD
		13	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: hh:mm:ss
		14	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm:ss
		15	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS): Global activation status				
	330	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
		1	-	GPS: Basic rotation 0 = Off, 1 = On
		3	Axis	GPS: Mirroring 0 = Off, 1 = On Index: 1 - 6 (X, Y, Z, A, B, C)
		4	-	GPS: Shift in the modified workpiece system 0 = Off, 1 = On
		5	-	GPS: Rotation in input system 0 = Off, 1 = On
		6	-	GPS: Feed rate factor 0 = Off, 1 = On
		8	-	GPS: Handwheel superimpositioning 0 = Off, 1 = On
		10	-	GPS: Virtual tool axis VT 0 = Off, 1 = On
		15	-	GPS: Selection of the handwheel coordinate system 0 = Machine coordinate system M-CS 1 = Workpiece coordinate system W-CS 2 = Modified workpiece coordinate system mW-CS 3 = Working plane coordinate system WPL-CS
		16	-	GPS: Shift in the workpiece system 0 = Off, 1 = On
		17	-	GPS: Axis offset 0 = Off, 1 = On

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 - 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 - 6 (X, Y, Z, A, B, C)
		5	-	GPS: Angle of rotation in input coordinate system I-CS
		6	-	GPS: Feed rate factor
		8	Axis	GPS: Handwheel superimpositioning Maximum value Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		9	Axis	GPS: Value for handwheel superimpositioning Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 - 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 - 6 (A, B, C)
TS touch trigger probe				
	350	50	1	Touch probe type: 0: TS120, 1: TS220, 2: TS440, 3: TS630, 4: TS632, 5: TS640, 6: TS444, 7: TS740
			2	Line in the touch-probe table
		51	-	Effective length
		52	1	Effective radius of the stylus tip
			2	Rounding radius
		53	1	Center offset (reference axis)
			2	Center offset (minor axis)
		54	-	Spindle-orientation angle in degrees (center offset)
		55	1	Rapid traverse
			2	Measuring feed rate
			3	Feed rate for pre-positioning: FMAX_PROBE or FMAX_MACHINE
		56	1	Maximum measuring range
			2	Set-up clearance
		57	1	Spindle orientation possible 0=No, 1=Yes
			2	Angle of spindle orientation in degrees

Group name	Group number ID...	System data number NO....	Index IDX...	Description
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
		71	1/2/3	TT: Touch probe center (REF system)
		72	-	TT: Touch probe radius
		75	1	TT: Rapid traverse
			2	TT: Measuring feed rate with stationary spindle
			3	TT: Measuring feed rate with rotating spindle
		76	1	TT: Maximum probing path
			2	TT: Safety clearance for linear measurement
			3	TT: Safety clearance for radius measurement
			4	TT: Distance from the lower edge of the cutter to the upper edge of the stylus
		77	-	TT: Spindle speed
		78	-	TT: Probing direction
		79	-	TT: Activate radio transmission
		80	-	TT: Stop probing movement upon stylus deflection
Preset from touch probe cycle (probing results)				
	360	1	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (input coordinate system). Compensations: length, radius, and center offset
		2	Axis	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (machine coordinate system, only axes from the active 3-D kinematics are allowed as index). Compensation: only center offset
		3	Coordinate	Result of measurement in the input system of touch probe Cycles 0 and 1. The measurement result is read out in the form of coordinates. Compensation: only center offset
		4	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (workpiece coordinate system) The measurement result is read in the form of coordinates. Compensation: only center offset
		5	Axis	Axis values, not compensated
		6	Coordinate / axis	Readout of the measurement results in the form of coordinates / axis values in the input system from probing operations. Compensation: only length
		10	-	Oriented spindle stop

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		11	-	Error status of probing: 0: Probing was successful –1: Touch point not reached –2: Touch probe already deflected at the start of the probing process
Read values from or write values to the active datum table				
	500	Row number	Column	Read values
Read values from or write values to the preset table (basic transformation)				
	507	Row number	1-6	Read values
Read axis offsets from or write axis offsets to the preset table				
	508	Row number	1-9	Read values
Data for pallet machining				
	510	1	-	Active line
		2	-	Current pallet number. Read value of the NAME column of the last PAL-type entry If the column is empty or does not contain a numerical value, a value of –1 is returned.
		3	-	Active row of the pallet table.
		4	-	Last line of the NC program for the current pallet.
		5	Axis	Tool-oriented editing: Clearance height is programmed: 0 = No, 1 = Yes Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		6	Axis	Tool-oriented editing: Clearance height The value is invalid if ID510 NR5 returns the value 0 with the corresponding IDX. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		10	-	Row number up to which the pallet table is to be searched during block scan.
		20	-	Type of pallet editing? 0 = Workpiece-oriented 1 = Tool oriented
		21	-	Automatic continuation after NC error: 0 = Locked 1 = Active 10 = Abort continuation 11 = Continuation with the rows in the pallet table that would have been executed next if not for the NC error 12 = Continuation with the row in the pallet table in which the NC error arose 13 = Continuation with the next pallet

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data from the point table				
	520	Row number	10	Read value from active point table.
			11	Read value from active point table.
			1-3 X/Y/Z	Read value from active point table.
Read or write the active preset				
	530	1	-	Number of the active preset in the active preset table.
Active pallet preset				
	540	1	-	Number of the active pallet preset. Returns the number of the active preset. If no pallet preset is active, the function returns the value –1.
		2	-	Number of the active pallet preset. As with NR1.
Values for the basic transformation of the pallet preset				
	547	row number	Axis	Read values of the basic transformation from the pallet preset table.. Index: 1 to 6 (X, Y, Z, SPA, SPB, SPC)
Axis offsets from the pallet preset table				
	548	Row number	Offset	Read values of the axis offsets from the pallet preset table.. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset.. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Read and write the machine status				
	590	2	1-30	Freely available; not deleted during program selection.
		3	1-30	Freely available; not deleted during a power failure (persistent storage).
Read/write look-ahead parameter of a single axis (at machine level)				
	610	1	-	Minimum feed rate (**MP_minPathFeed**) in mm/min
		2	-	Minimum feed rate at corners (**MP_minCornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPathJerk**) in m/s^3
		5	-	Max. jerk at high speeds (**MP_maxPathJerkHi**) in m/s^3
		6	-	Tolerance at low speeds (**MP_pathTolerance**) in mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tolerance at high speeds (**MP_pathToleranceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPathYank**) in m/s^4
		9	-	Tolerance factor for curve machining (**MP_curveTolFactor**)
		10	-	Factor for max. permissible jerk at curvature changes (**MP_curveJerkFactor**)
		11	-	Maximum jerk with probing movements (**MP_pathMeasJerk**)
		12	-	Angle tolerance for machining feed rate (**MP_angleTolerance**)
		13	-	Angle tolerance for rapid traverse (**MP_angleToleranceHi**)
		14	-	Max. corner angle for polygons (**MP_maxPolyAngle**)
		18	-	Radial acceleration with machining feed rate (**MP_maxTransAcc**)
		19	-	Radial acceleration with rapid traverse (**MP_maxTransAccHi**)
		20	Index of physical axis	Max. feed rate (**MP_maxFeed**) in mm/min
		21	Index of physical axis	Max. acceleration (**MP_maxAcceleration**) in m/s^2
		22	Index of physical axis	Maximum transition jerk of the axis in rapid traverse (**MP_axTransJerkHi**) in m/s^2
		23	Index of physical axis	Maximum transition jerk of the axis during machining free rate (**MP_axTransJerk**) in m/s^3
		24	Index of physical axis	Acceleration feedforward control (**MP_compAcc**)
		25	Index of physical axis	Axis-specific jerk at low speeds (**MP_axPathJerk**) in m/s^3
		26	Index of physical axis	Axis-specific jerk at high speeds (**MP_axPathJerkHi**) in m/s^3
		27	Index of physical axis	More precise tolerance examination in corners (**MP_reduceCornerFeed**) 0 = deactivated, 1 = activated
		28	Index of physical axis	DCM: Maximum tolerance for linear axes in mm (**MP_maxLinearTolerance**)
		29	Index of physical axis	DCM: Maximum angle tolerance in [°] (**MP_maxAngleTolerance**)
		30	Index of physical axis	Tolerance monitoring for successive threads (**MP_threadTolerance**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		31	Index of physical axis	Form (**MP_shape**) of the **axisCutterLoc** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		32	Index of physical axis	Frequency **MP_frequency**) of the **axisCutterLoc** filter in Hz
		33	Index of physical axis	Form (**MP_shape**) of the **axisPosition** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		34	Index of physical axis	Frequency (**MP_frequency**) of the **axisPosition** filter in Hz
		35	Index of physical axis	Order of the filter for **Manual** operating mode (**MP_manualFilterOrder**)
		36	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisCutterLoc** filter
		37	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisPosition** filter
		38	Index of physical axis	Axis-specific jerk for probing movements (**MP_axMeasJerk**)
		39	Index of physical axis	Weighting of the filter error for calculating filter deviation (**MP_axFilterErrWeight**)
		40	Index of physical axis	Maximum filter length of position filter (**MP_maxHscOrder**)
		41	Index of physical axis	Maximum filter length of CLP filter (**MP_maxHscOrder**)
		42	-	Maximum feed rate of the axis at machining feed rate (**MP_maxWorkFeed**)
		43	-	Maximum path acceleration at machining feed rate (**MP_maxPathAcc**)
		44	-	Maximum path acceleration at rapid traverse (**MP_maxPathAccHi**)
		51	Index of physical axis	Compensation of following error in the jerk phase (**MP_IpcJerkFact**)
		52	Index of physical axis	kv factor of the position controller in 1/s (**MP_kvFactor**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Measure the maximum utilization of an axis				
	621	0	Index of physical axis	Conclude measurement of the dynamic load and save the result in the specified Q parameter.
Read SIK contents				
	630	0	Option no.	You can explicitly determine whether the SIK option given under **IDX** has been set or not. 1 = option is enabled 0 = option is not enabled
		1	-	You can determine whether a Feature Content Level (for upgrade functions) is set, and which one. –1 = No FCL is set <No.> = FCL that is set
		2	-	Read serial number of the SIK -1 = No valid SIK in the system
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)
Counter				
	920	1	-	Planned workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		2	-	Already machined workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		12	-	Workpieces still to be machined. In **Test Run** operating mode the counter generally generates the value 0.
Read and write data of current tool				
	950	1	-	Tool length L
		2	-	Tool radius R
		3	-	Tool radius R2
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Tool radius oversize DR2
		7	-	Tool locked TL 0 = not locked, 1 = locked
		8	-	Number of the replacement tool RT
		9	-	Maximum tool age TIME1
		10	-	Maximum tool age TIME2 at TOOL CALL
		11	-	Current tool age CUR.TIME
		12	-	PLC status

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		13	-	Tooth length in the tool axis LCUTS
		14	-	Maximum plunge angle ANGLE
		15	-	TT: Number of tool teeth CUT
		16	-	TT: Wear tolerance for length LTOL
		17	-	TT: Wear tolerance for radius RTOL
		18	-	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	-	TT: Offset in plane R-OFFS R = 99999.9999
		20	-	TT: Offset in length L-OFFS
		21	-	TT: Break tolerance for length LBREAK
		22	-	TT: Break tolerance for radius RBREAK
		28	-	Maximum spindle speed [rpm] NMAX
		32	-	Point angle TANGLE
		34	-	LIFTOFF allowed (0 = No, 1 = Yes)
		35	-	Wear tolerance for radius R2TOL
		36	-	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	-	Corresponding line in the touch-probe table
		38	-	Timestamp of last use
		39	-	ACC
		40	-	Pitch for thread cycles
		44	-	Exceeding the tool life

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Freely available memory area for tool management				
	956	0-9	-	Freely available data area for tool management. The data is not reset when the program is aborted.
Tool usage and tooling				
	975	1	-	Tool usage test for the current NC program: Result –2: Test not possible, function disabled in the configuration Result –1: Test not possible, tool usage file missing Result 0: Test OK, all tools available Result 1: Test not OK
		2	Line	Check availability of the tools required in the pallet from line IDX in the current pallet table. –3 = No pallet is defined in row IDX, or function was called outside of pallet editing –2 / –1 / 0 / 1 see NR1
Lift off the tool at NC stop				
	980	3	-	(This function is obsolete—HEIDENHAIN recommends not to use it any longer. ID980 NR3 = 1 is equivalent to ID980 NR1 = –1, ID980 NR3 = 0 has the same effect as ID980 NR1 = 0. Other values are not permissible.) Enable lift-off to the value defined in CfgLiftOff: 0 = Lock lift-off function 1 = Enable lift-off function
Touch probe cycles and coordinate transformations				
	990	1	-	Approach behavior: 0 = Standard behavior 1 = Approach probing position without compensation Effective radius, set-up clearance is zero
		2	16	Automatic / Manual machine operating modes
		4	-	0 = Stylus not deflected 1 = Stylus deflected
		6	-	TT tool touch probe active? 1 = Yes 0 = No
		8	-	Momentary spindle angle in [°]
		10	QS parameter no.	Determine the tool number from the tool name. The return value depends on the rules configured for the search of the replacement tool. If there are multiple tools with the same name, the first tool from the tool table will be selected.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				If the tool selected by these rules is locked, a replacement tool will be returned. –1: No tool with the specified name found in the tool table or all qualifying tools are locked.
		16	0	0 = Transfer control over the channel spindle to the PLC, 1 = Assume control over the channel spindle
			1	0 = Pass tool spindle control to the PLC, 1 = Take control of the tool spindle
		19	-	Suppress touch prove movement in cycles: 0 = Movement will be suppressed (CfgMachineSimul/simMode parameter not equal to FullOperation or **Test Run** operating mode is active) 1 = Movement will be performed (CfgMachineSimul/simMode parameter = FullOperation, can be programmed for testing purposes)
Status of execution				
	992	10	-	Block scan active 1 = yes, 0 = no
		11	-	Block scan—information on block scan: 0 = NC program started without block scan 1 = Iniprog system cycle is run before block scan 2 = Block scan is running 3 = Functions are being updated –1 = Iniprog cycle was canceled before block scan –2 = Cancellation during block scan –3 = Cancellation of the block scan after the search phase, before or during the update of functions –99 = Implicit cancellation
		12	-	Type of canceling for interrogation within the OEM_CANCEL macro: 0 = No cancellation 1 = Cancellation due to error or emergency stop 2 = Explicit cancellation with internal stop after stop in the middle of the block 3 = Explicit cancellation with internal stop after stop at the end of a block
		14	-	Number of the last FN14 error
		16	-	Real execution active? 1 = execution, 0 = simulation
		17	-	2-D graphics during programming active? 1 = yes 0 = no

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		18	-	Live programming graphics (**AUTO DRAW** soft key) active? 1 = Yes 0 = No
		20	-	Information on combined milling/turning mode of operation: 0 = Milling (after **FUNCTION MODE MILL**) 1 = Turning (after **FUNCTION MODE TURN**) 10 = Execute the operations for the turning-to-milling transition 11 = Execute the operations for the milling-to-turning transition
		30	-	Interpolation of multiple axes permitted? 0 = No (e.g. for straight cut control) 1 = yes
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes
		32	0	Cycle call possible/permitted? 0 = No 1 = Yes
			Cycle number	Single cycle enabled: 0 = No 1 = Yes
		40	-	Copy tables in **Test Run** operating mode? Value 1 will be set when a program is selected and when the **RESET+START** soft key is pressed. The **iniprog.h** system cycle will then copy the tables and reset the system datum. 0 = no 1 = yes
		101	-	M101 active (visible condition)? 0 = no 1 = yes
		136	-	M136 active? 0 = no 1 = yes

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Activate machine parameter subfile				
	1020	13	QS parameter no.	Has a machine parameter subfile with path from QS number (IDX) been loaded? 1 = Yes 0 = No
Configuration settings for cycles				
	1030	1	-	Display **spindle does not rotate** error message? **(CfgGeoCycle/displaySpindleErr)** 0 = no, 1 = yes
			-	Check **the algebraic sign for depth error message!** display? **(CfgGeoCycle/displayDepthErr)** 0 = no, 1 = yes
Write or read PLC data synchronously in real time				
	2000	10	Marker no.	PLC markers General note for NR10 to NR80: The functions are executed synchronously in real time, i.e. the function is not executed until the corresponding point is reached in the program. HEIDENHAIN recommends using the **WRITE TO PLC** or **READ FROM PLC** commands instead of ID2000 and synchronizing the execution in real time by using **FN20: WAIT FOR SYNC**.
		20	Input no.	PLC input
		30	Output no.	PLC output
		40	Counter no.	PLC counter
		50	Timer no.	PLC timer
		60	Byte no.	PLC byte
		70	Word no.	PLC word
		80	Double-word no.	PLC double word

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Do not write or read PLC data synchronously in real time				
	2001	10-80	see ID 2000	Same as ID2000 NR10 to NR80, but not synchronous in real time. Function is executed in the look-ahead calculation. HEIDENHAIN recommends using the **WRITE TO PLC** and **READ FROM PLC** commands instead of ID2001.
Bit test				
	2300	Number	Bit number	This function checks whether a bit has been set in a number. The number to be checked is transferred as NR, the bit to be searched for as IDX, with IDX0 designating the least significant bit. To call this function for great numbers, make sure to transfer NR as a Q parameter. 0 = Bit not set 1 = Bit set
Read program information (system string)				
	10010	1	-	Path of the current main program or pallet program.
		2	-	Path of the NC program shown in the block display.
		3	-	Path of the cycle selected with **SEL CYCLE** or **CYCLE DEF 12 PGM CALL**, or path of the currently active cycle
		10	-	Path of the NC program selected with **SEL PGM "..."**.
Read channel data (system string)				
	10025	1	-	Name of machining channel (key)
Read data for SQL tables (system string)				
	10040	1	-	Symbolic name of the preset table.
		2	-	Symbolic name of the datum table.
		3	-	Symbolic name of the pallet preset table.
		10	-	Symbolic name of the tool table.
		11	-	Symbolic name of the pocket table.
		12	-	Symbolic name of the turning tool table

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed in the tool call (system string)				
	10060	1	-	Tool name
Read machine kinematics (system strings)				
	10290	10	-	Symbolic name of the machine kinematics from Channels/ChannelSettings/CfgKinList/kinCompositeModels programmed in **FUNCTION MODE MILL** or **FUNCTION MODE TURN**.
Traverse range switchover (system string)				
	10300	1	-	Key name of the last active range of traverse
Read current system time (system string)				
	10321	1 - 16	-	1: DD.MM.YYYY hh:mm:ss 2 and 16: DD.MM.YYYY hh:mm 3: DD.MM.YY hh:mm 4: YYYY-MM-DD hh:mm:ss 5 and 6: YYYY-MM-DD hh:mm 7: YY-MM-DD hh:mm 8 and 9: DD.MM.YYYY 10: DD.MM.YY 11: YYYY-MM-DD 12: YY-MM-DD 13 and 14: hh:mm:ss 15: hh:mm As an alternative, you can use **DAT** in **SYSSTR(...)** to specify a system time in seconds that is to be used for formatting.
Read data of touch probes (TS, TT) (system string)				
	10350	50	-	TS probe type from TYPE column of the touch probe table (**tchprobe.tp**)
		70	-	Type of TT tool touch probe from CfgTT/type.
		73	-	Key name of the active tool touch probe TT from **CfgProbes/activeTT**.
Read and write data of touch probes (TS, TT) (system string)				
	10350	74	-	Serial number of the active tool touch probe TT from **CfgProbes/activeTT**.
Read the data for pallet machining (system string)				
	10510	1	-	Pallet name
		2	-	Path of the selected pallet table.
Read version ID of the NC software (system string)				
	10630	10	-	The string corresponds to the format of the version ID shown, e.g. **340590 09** or **817601 05 SP1**.
Read information on unbalance cycle (system string)				
	10855	1	-	Path of the unbalance calibration table belonging to the active kinematics

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data of the current tool (system string)				
	10950	1	-	Current tool name
		2	-	Entry from the DOC column of the active tool
		3	-	AFC control setting
		4	-	Tool-carrier kinematics
		5	-	Entry from the DR2TABLE column – file name of the compensation value table for 3D-ToolComp

Comparison: FN 18 functions

The following table lists the FN18 functions from previous controls, which were not implemented in this manner in the TNC 128.

In most cases, this function has been replaced by another function.

No.	IDX	Contents	Replacement function
ID 10 Program information			
1	-	mm/inch condition	Q113
2	-	Overlap factor for pocket milling	CfgRead
4	-	Number of the active fixed cycle	ID 10 no. 3
ID 20 Machine status			
15	Log. axis	Assignment between logic and geometric axes	
16	-	Feed rate for transition arcs	
17	-	Currently selected range of traverse	SYSTRING 10300
19	-	Maximum spindle speed for current gear stage and spindle	Maximum gear range: ID 90 No. 2
ID 50 Data from the tool table			
23	Tool no.	PLC value	1)
24	Tool no.	Probe center offset in reference axis (CAL-OF1)	ID 350 NR 53 IDX 1
25	Tool no.	Probe center offset in minor axis (CALOF-2)	ID 350 NR 53 IDX 2
26	Tool no.	Spindle angle during calibration (CAL-ANG)	ID 350 NR 54
27	Tool no.	Tool type for pocket table (PTYP)	2)
29	Tool no.	Position P1	1)
30	Tool no.	Position P2	1)
31	Tool no.	Position P3	1)
33	Tool no.	Thread pitch (Pitch)	ID 50 NR 40
ID 51 Data from the pocket table			
6	Pocket no.	Tool type	2)
7	Pocket no.	P1	2)
8	Pocket no.	P2	2)

No.	IDX	Contents	Replacement function
9	Pocket no.	P3	2)
10	Pocket no.	P4	2)
11	Pocket no.	P5	2)
12	Pocket no.	Pocket reserved 0 = No, 1 = Yes	2)
13	Pocket no.	Box magazine: Pocket above occupied: 0 = No, 1 = Yes	2)
14	Pocket no.	Box magazine: Pocket below occupied: 0 = No, 1 = Yes	2)
15	Pocket no.	Box magazine: Pocket to the left occupied: 0 = No, 1 = Yes	2)
16	Pocket number	Box magazine: Pocket to the right occupied: 0 = No, 1 = Yes	2)
ID 56 File information			
1	-	Number of lines of the tool table	
2	-	Number of lines of the active datum table	
3	Q parameters	Number of active axes that are programmed in the active datum table	
4	-	Number of lines in a freely definable table that has been opened with FN26: TABOPEN	
ID 214 Current contour data			
1	-	Contour transition mode	
2	-	Max. linearization error	
3	-	Mode for M112	
4	-	Character mode	
5	-	Mode for M124	1)
6	-	Specification for contour pocket machining	
7	-	Filter for control loop	
8	-	Tolerance programmed with Cycle 32 or MP 1096	ID 30 no. 48
ID 240 Nominal positions in the REF system			
8	-	ACTUAL position in the REF system	
ID 280 Information on M128			
2	-	Feed rate that was programmed with M128	ID 280 NR 3
ID 290 Switch the kinematics			
1	-	Line of the active kinematics table	SYSSTRING 10290
2	Bit no.	Interrogate the bits in MP7500	Cfgread
3	-	Status of collision monitoring (old)	Can be activated and deactivated in the NC program

No.	IDX	Contents	Replacement function
4	-	Status of collision monitoring (new)	Can be activated and deactivated in the NC program
ID 310 Modifications of geometrical behavior			
116	-	M116: –1 = On, 0 = Off	
126	-	M126: –1 = On, 0 = Off	
ID 350 Touch-probe data			
10	-	TS: Touch-probe axis	ID 20 NR 3
11	-	TS: Effective ball radius	ID 350 NR 52
12	-	TS: Effective length	ID 350 NR 51
13	-	TS: Ring gauge radius	
14	1/2	TS: Center offset in reference/minor axis	ID 350 NR 53
15	-	TS: Direction of center offset relative to 0° position	ID 350 NR 54
20	1/2/3	TT: Center point X/Y/Z	ID 350 NR 71
21	-	TT: Plate radius	ID 350 NR 72
22	1/2/3	TT: 1st probing position X/Y/Z	Cfgread
23	1/2/3	TT: 2nd probing position X/Y/Z	Cfgread
24	1/2/3	TT: 3rd probing position X/Y/Z	Cfgread
25	1/2/3	TT: 4th probing position X/Y/Z	Cfgread
ID 370 Touch probe cycle settings			
1	-	Do not move to set-up clearance in Cycle 0.0 and 1.0 (as with ID990 NR1)	ID 990 NR 1
2	-	MP 6150 Rapid traverse for measurement	ID 350 NR 55 IDX 1
3	-	MP 6151 Machine rapid traverse as rapid traverse for measurement	ID 350 NR 55 IDX 3
4	-	MP 6120 Feed rate for measurement	ID 350 NR 55 IDX 2
5	-	MP 6165 Angle tracking on/off	ID 350 NR 57
ID 501 Datum table (REF system)			
Line	Column	Value in datum table	Preset table
ID 502 Preset table			
Line	Column	Read the value from preset table, taking into account the active machining system	
ID 503 Preset table			
Line	Column	Read the value directly from the preset table	ID 507
ID 504 Preset table			
Line	Column	Read the basic rotation from the preset table	ID 507 IDX 4-6
ID 505 Datum table			
1	-	0 = No datum table selected 1 = Datum table selected	

No.	IDX	Contents	Replacement function
ID 510 Data for pallet machining			
7	-	Test the insertion of a fixture from the PAL line	
ID 530 Active preset			
2	Line	Write-protect the line in the active preset table: 0 = No, 1 = Yes	FN 26/28 Read out the Locked column
ID 990 Approach behavior			
2	10	0 = No execution in block scan 1 = Execution in block scan	ID 992 NR 10 / NR 11
3	Q parameters	Number of axes that are programmed in the selected datum table	
ID 1000 Machine parameter			
MP number	MP index	Value of the machine parameter	CfgRead
ID 1010 Machine parameter is defined			
MP number	MP index	0 = Machine parameter does not exist 1 = Machine parameter exists	CfgRead

1) Function or table column no longer exists

2) Use FN 26 / FN 28 or SQL to read out the table cell

18.2 Technical Information

Specifications

Explanation of symbols

- ■ Default
- □ Axis option
- **1** Advanced Function Set 1

Specifications		
Components	■	Operating panel
	■	Screen with soft keys
Program memory	■	2 GB
Input resolution and display step	■	As fine as 0.1 µm for linear axes
	■	Up to 0.0001° for rotary axes
Input range	■	Maximum 999 999 999 mm or 999 999 999°
Block processing time	■	6 ms
Axis feedback control	■	Position loop resolution: Signal period of the position encoder/1024
	■	Cycle time of position controller: 3 ms
	■	Cycle time of speed controller: 200 µs
Range of traverse	■	Maximum 100 m (3937 inches)
Spindle speed	■	Maximum 100,000 rpm (analog speed command signal)
Error compensation	■	Linear and nonlinear axis error, backlash, thermal expansion
	■	Static friction
Data interfaces	■	One each RS-232-C /V.24 max. 115 kilobaud
	■	Expanded data interface with LSV-2 protocol for remote operation of the control through the data interface with the **TNCremo** software
	■	Ethernet interface 1000 BaseT
	■	3 x USB (1 x front USB 2.0; 2 x rear USB 3.0)
Ambient temperature	■	Operation: 5 °C to +45 °C
	■	Storage: –35 °C to +65 °C

Input formats and units of control functions	
Positions, coordinates, chamfer lengths	–99 999.9999 to +99 999.9999 (5,4: number of digits before and after the decimal point) [mm]
Tool numbers	0 to 32 767.9 (5, 1)
Tool names	32 characters, enclosed by quotation marks with **TOOL CALL**. Permitted special characters: # $ % & . , - _
Detail values for tool compensation	–99.9999 to +99.9999 (2, 4) [mm]
Spindle speeds	0 to 99 999.999 (5, 3) [rpm]
Feed rates	0 to 99,999.999 (5, 3) [mm/min] or [mm/tooth] or [mm/1]
Dwell time in Cycle 9	0 to 3600.000 (4, 3) [s]
Thread pitch in various cycles	-9.9999 to +9.9999 (2, 4) [mm]
Angle for spindle orientation	0 to 360.0000 (3, 4) [°]
Datum numbers in Cycle 7	0 to 2999 (4, 0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2, 6)
Miscellaneous functions M	0 to 999 (4, 0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	–99 999.9999 to +99 999.9999 (9, 6)
Labels (LBL) for program jumps	0 to 999 (5, 0)
Labels (LBL) for program jumps	Any text string in quotation marks (“ ”)
Number of program section repeats REP	1 to 65 534 (5, 0)
Error number for Q parameter function FN 14	0 to 1199 (4, 0)

User functions

User functions	
Short description	■ Basic version: 3 axes plus closed-loop spindle □ 1. Additional axis for 4 axes plus closed-loop spindle □ 2. Additional axis for 5 axes plus closed-loop spindle
Program entry	In HEIDENHAIN conversational format
Position entry	■ Nominal positions for straight lines in Cartesian coordinates ■ Incremental or absolute dimensions ■ Display and entry in mm or inches
Tool tables	Multiple tool tables with any number of tools
Parallel operation	Creating an NC program with graphical support while another NC program is being run
Cutting data	Automatic calculation of spindle speed, cutting speed, feed per tooth and feed per revolution
Program jumps	■ Subprograms ■ Program section repeats ■ Any desired NC program as subprogram
Machining cycles	■ Cycles for drilling, and conventional and rigid tapping ■ Roughing and finishing rectangular pockets ■ Cycles for pecking, reaming, boring, and counterboring ■ Roughing and finishing rectangular studs ■ Cycles for clearing level surfaces ■ Face milling ■ Cartesian and polar point patterns ■ OEM cycles (special cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	■ Datum shift, mirroring ■ Scaling factor (axis-specific)
Q parameters Programming with variables	■ Mathematical functions =, +, –, *, /, roots ■ Logical operations (=, ≠, <, >) ■ Calculating with parentheses ■ sin α, cos α, tan α , arc sin, arc cos, arc tan, a^n, e^n, In, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point ■ Functions for calculation of circles ■ String param.
Programming aids	■ Calculator ■ Complete list of all current error messages ■ Context-sensitive help function for error messages ■ TNCguide: The integrated help system. ■ Graphic support for the programming of cycles ■ Comment and structure blocks in the NC program

User functions		
Teach-In	■	Actual positions can be transferred directly to the NC program
Test graphics Display modes	■ ■ ■	Graphic simulation before a program run, even while another NC program is being run Plan view / projection in 3 planes / 3-D view Detail enlargement
Programming graphics	■	In the **Programming** operating mode, the contours of the NC blocks are drawn on screen while they are being entered (2-D pencil-trace graphics), even while another NC program is being run
Program-run graphics Display modes	■	Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■ ■	Calculation of machining time in the **Test Run** operating mode Display of the current machining time in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes
Preset management	■	For saving any presets
Contour, returning to	■ ■	Block scan in any NC block in the NC program, returning the tool to the calculated nominal position to continue machining NC program interruption, contour departure and return
Datum tables	■	Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	■ ■ ■	Calibrating the touch probe Presetting, manual Tools can be measured automatically

Software options

Touch Probe Functions (option 17)	
Touch probe functions	**Touch probe cycles:** ■ Presetting in the **Manual operation** mode ■ Tools can be measured automatically

HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component

Accessories

Accessories	
Electronic handwheels	■ HR 510: Portable handwheel ■ HR 550FS: Portable radio handwheel with display ■ HR 520: Portable handwheel with display ■ HR 420: Portable handwheel with display ■ HR 130: Panel-mounted handwheel ■ HR 150: Up to three panel-mounted handwheels via handwheel adapter HRA 110
Touch probes	■ TS 248: 3-D touch trigger probe with cable connection ■ TS 260: 3-D touch trigger probe with cable connection ■ TT 160: 3-D touch trigger probe for tool measurement ■ KT 130: Simple touch trigger probe with cable connection

Fixed cycles

Cycle number	Cycle name	DEF active	CALL active
7	DATUM SHIFT	■	
8	MIRROR IMAGE	■	
9	DWELL TIME	■	
11	SCALING	■	
12	PGM CALL		■
13	ORIENTATION	■	
200	DRILLING		■
201	REAMING		■
202	BORING		■
203	UNIVERSAL DRILLING		■
204	BACK BORING		■
205	UNIVERSAL PECKING		■
206	TAPPING		■
207	RIGID TAPPING		■
220	POLAR PATTERN	■	
221	CARTESIAN PATTERN	■	
233	FACE MILLING		■
240	CENTERING		■
241	SINGLE-LIP D.H.DRLNG		■
247	PRESETTING	■	
251	RECTANGULAR POCKET		■
253	SLOT MILLING		■
256	RECTANGULAR STUD		■

Miscellaneous functions

M	Effect	Effective at block Start	End	Page
M0	Program STOP/Spindle STOP/Coolant OFF		■	153
M1	Optional program run STOP/Spindle STOP/Coolant OFF		■	153
M2	Stop program/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 1		■	153
M3 M4 M5	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	■ ■	 ■	153
M6	Tool change/STOP program run (depending on machine parameter)/Spindle STOP		■	153
M8 M9	Coolant ON Coolant OFF	■	 ■	153
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant on	■ ■		153
M30	Same function as M2		■	153
M89	Vacant miscellaneous function **or** cycle call, modally effective (depending on machine parameter)	■	 ■	290
M91	Within the positioning block: Coordinates are referenced to machine datum	■		154
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position	■		154
M94	Reduce the rotary axis display to a value below 360°	■		156
M99	Blockwise cycle call		■	290
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136	■		158
M140	Retraction from the contour in the tool-axis direction	■		158

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.de

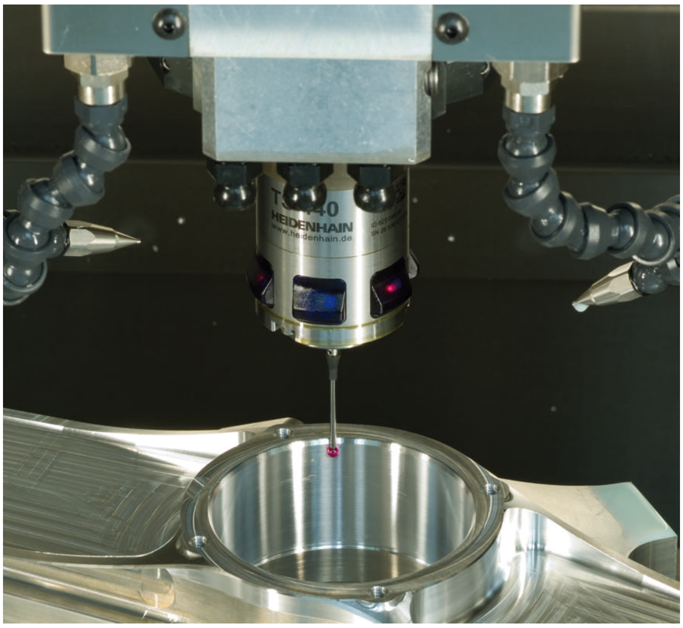
Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220	Signal transmission by cable
TS 440, TS 444	Infrared transmission
TS 640, TS 740	Infrared transmission

- Workpiece alignment
- Setting presets
- Workpiece measurement



Tool touch probes

TT 140	Signal transmission by cable
TT 449	Infrared transmission
TL	Non-contacting laser systems

- Tool measurement
- wear monitoring
- tool breakage detection

Original document
819494-25 · Ver05 · SW07 · 10/2018 · H · Printed in Germany