HEIDENHAIN

TNC 128

User's Manual
Klartext Programming

NC Software
771841-18

English (en)
10/2023

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

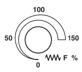
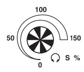
Feed rate	Spindle speed
	

Table of contents

1

Basic Information

1.1 About this manual

Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer'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 property damage**.

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

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

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

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

> The book symbol indicates a **cross reference**.
>
> A cross reference leads to external documentation for example the documentation of your machine manufacturer 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.

> HEIDENHAIN has simplified the version schema, starting with NC software version 16:
> - The publication period determines the version number.
> - All control models of a publication period have the same version number.
> - The version number of the programming stations corresponds to the version number of the NC software.

Control model	NC software number
TNC 128	771841-18
TNC 128 Programming Station	771845-18

The machine manufacturer adapts the usable features of the control to his machine by setting appropriate 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 3D 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, each of which can be enabled separately by your machine manufacturer. The respective options provide the functions listed below:

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:** ■ Set the preset in the **Manual operation** mode of operation ■ Tools can be measured automatically
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component

Further options available

HEIDENHAIN offers further hardware enhancements and software options that can be configured and implemented only by your machine manufacturer.

For more information, please refer to your machine manufacturer's documentation or the HEIDENHAIN brochure titled **Options and Accessories**.

ID: 827222-xx

VTC User's Manual

All functions of the software for the VT 121 vision system are described in the **VTC User's Manual**. Please contact HEIDENHAIN if you require a copy of this User's Manual.

ID: 1322445-xx

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

The control software contains open-source software that is subject to special terms of use. These special terms of use have priority.

To call further information on the control:

- Press the **MOD** key
- Select the **General Information** group in the MOD menu
- Select the **License information** MOD function

When using the OPC UA NC server or DNC server, you can influence the behavior of the control. Therefore, before using these interfaces for productive purposes, verify that the control can still be operated without malfunctions or drops in performance. The manufacturer of the software that uses these communication interfaces is responsible for performing system tests.

New and modified functions with 77184x-18

Overview of new and modified software functions

Further information about the previous software versions is presented in the **Overview of New and Modified Software Functions** documentation. Please contact HEIDENHAIN if you need this documentation.

ID: 1322088-xx

Further information: User's Manual for **Programming of Machining Cycles**

- The **Display Step** software option (option 23) is available in the standard control version. The display step of the axes is no longer limited to four decimal places.

  The machine parameter **displayPace** (no. 101000) allows you to define the display step for the individual axes. The minimum display step of the axes is 0.1 µm or 0.0001°.
- **State Reporting Interface** (software option 137) is no longer available.

New functions

- **FUNCTION CORRDATA** allows you to activate a line of the compensation table. The compensation remains effective until the next tool change or the end of the program.

  Further information: "Activate the compensation table", Page 330
- **FUNCTION MODE SET** allows you to activate settings defined by the machine manufacturer (e.g., changes to the range of traverse) from within the NC program

  Further information: "Function Mode Set", Page 291
- The function **PRESET SELECT** allows you to activate a preset from the preset table. You can choose to retain active transformations and select the preset to which the function should apply.

  Further information: "Activating a preset", Page 319
- The function **PRESET COPY** allows you to copy a preset defined in the preset table to another line. You can optionally activate the copied preset and retain the active transformations.

  Further information: "Copying a preset", Page 321
- The function **PRESET CORR** corrects the active preset.

  Further information: "Correcting a preset", Page 322
- The function **OPEN FILE** allows the control to open different file types (e.g., PNG files) using a suitable additional tool.

  Further information: "OPEN FILE", Page 308

- The function **TABDATA** allows you to access the tool table and the compensation tables (*.tco and *.wco) during program run. In order to access the compensation tables, you need to activate them.
  - The function **TABDATA READ** allows you to read a value from a table and save it to a Q, QL, QR, or QS parameter.
  - The function **TABDATA WRITE** allows you to write a value from a Q, QL, QR, or QS parameter into a table.
  - The function **TABDATA ADD** allows you to add a value from a Q, QL, or QR parameter to a value contained in a table.

  Further information: "Accessing table values ", Page 332
- The **SELECT FILE** soft key has been added to the selection window provided by the **APPLY FILE NAME** soft key. If the called file is located in the same directory as the file you are calling it from, pressing this soft key loads the name of the file without its path.
- The following NC functions for coordinate transformations have been added:
  - Use the **TRANS MIRROR** function to mirror contours or positions about one or more axes. The **TRANS MIRROR RESET** function allows you to reset mirroring. The NC functions serve as an alternative to Cycle **8 MIRRORING**.
  - The **TRANS SCALE** function lets you change the scale of the contours or distances to the datum, thereby evenly enlarging or shrinking them. This enables you to program shrinkage and oversize allowances, for example. Use the **TRANS SCALE RESET** function to reset scaling. The NC functions serve as an alternative to Cycle **11 SCALING FACTOR**.
  - Use the NC function **TRANS RESET** to reset all simple coordinate transformations simultaneously.

  Further information: "NC functions for coordinate transformations", Page 310
- During retraction with **M140 MB MAX**, the control takes into account the safety clearances that can be defined by the machine manufacturer for software limit switches and collision objects. The control reduces the retraction movements by the clearances and stops before the software limit switches.

  Further information: "Retraction from the contour in the tool-axis direction: M140", Page 174

- In the mask file of the **FN 16: F-PRINT** function, you can define whether the control shows or hides blank lines for undefined QS parameters.

  Further information: "FN 16: F-PRINT – Formatted output of text and Q parameter values", Page 233
- The function **SYSSTR(ID10321 NR20)** determines the number of the current week in accordance with ISO 8601.

  Further information: "Reading system data", Page 253
- Using the **SYNTAX** soft key, you can enclose path information in quotation marks in order to use any special characters as part of the path (e.g., **/**). The control provides the **SYNTAX** soft key with the following NC functions:
  - **FN 16: F-PRINT** (ISO: **D16**)
  - **FN 26: TABOPEN** (ISO: **D26**)
  - Cycle **12 PGM CALL** (ISO: **G39**)
  - **CALL PGM** (ISO: **%**)
- The **FN 18: SYSREAD** (ISO: **D18**) functions have been extended:
  - **FN 18: SYSREAD (D18) ID10**: Read program information
    - **NR10**: Counts the number of executions of the current program section
  - **FN 18: SYSREAD (D18) ID15**
    - **NR10**: contents of a Q parameter
    - **NR11**: contents of a QL parameter
    - **NR12**: contents of a QR parameter
  - **FN 18: SYSREAD (D18) ID35 NR2**: active radius compensation
  - **FN 18: SYSREAD (D18) ID50**: values in the tool table
    - **NR45**: value in the **RCUTS** column
    - **NR46**: value in the **LU** column
  - **FN 18: SYSREAD (D18) ID245 NR1**: Current nominal position of an axis (**IDX**) in the REF system
  - **FN 18: SYSREAD (D18) ID370 NR7**: Reaction of the control if a probing point is not reached during a programmable touch-probe cycle **14xx** (option 17)
  - **FN 18: SYSREAD (D18) ID630**: SIK information of the control
    - **NR3**: SIK generation **SIK1** or **SIK2**
    - **NR4**: Specifies whether and how often a software option (**IDX**) has been enabled on controls with **SIK2**
  - **FN 18: SYSREAD (D18) ID950**: tool-table values for the current tool
    - **NR45**: value in the **RCUTS** column
    - **NR46**: value in the **LU** column
    - **NR47**: value in the **RN** column
    - **NR48**: value in the **R_TIP** column
  - **FN 18: SYSREAD (D18) ID990 NR28**: Current tool spindle angle
  - **FN 18: SYSREAD (D18) ID1070 NR1**: active feed-rate limit through the **F MAX** soft key

- **FN 18: SYSREAD (D18) ID10010 NR1** and **NR2**: information about the current main program or the called NC program as a text variable
  - **IDX1**: directory path
  - **IDX2**: file name
  - **IDX3**: file type
- **FN 18: SYSREAD (D18) ID10015**
  - **NR20**: contents of a QS parameter
  - **NR30**: contents of a QS parameter (all characters, except the letters and numbers, are replaced with an underscore (_) character)

Further information: "System data", Page 518

- If you use the **SQL EXECUTE** function and the **CREATE TABLE** statement to create a table, then you define the sequence of the columns with the **AS SELECT** statement.

  Further information: "SQL EXECUTE", Page 269
- The **SELECT COMPENS. TABLE** soft key has been added to the soft-key row of the **PGM CALL** functions. This soft key activates the **SEL CORR-TABLE** function that allows you to activate a compensation table for the NC program.

  Further information: "Activate the compensation table", Page 330

- The control includes the sample tables **WMAT.tab**, **TMAT.tab** and **EXAMPLE.cutd** for automatic cutting data calculation.

  Further information: "Cutting data calculator", Page 146
- If, after a hardware change or an update, an error occurs when the control is booting, the control will automatically open the error window and display a question-type error. The control displays soft keys providing different response options.

  Further information: "Display of errors", Page 153
- In the error window, the **ACTIVATE AUTOMATIC SAVING** soft key has been added to **MORE FUNCTIONS**. This soft key allows you to define up to five error numbers. The control will automatically create a service file upon occurrence of these error numbers.

  Further information: "ACTIVATE AUTOMATIC SAVING soft key", Page 155
- The control saves active NC programs only up to a maximum size of 10 MB each to a service file. NC programs larger than that are not saved.

  Further information: "Saving service files", Page 159
- In the optional machine parameter **CfgClearError** (no. 130200), the machine manufacturer defines whether the control automatically clears warning and error messages when an NC program is selected or restarted.
- The CAD Viewer has been enhanced as follows:
  - In **CAD Viewer**, you can choose the **YZ** and **ZX** working planes for milling. You can choose the desired working plane from a selection menu.

  Further information: "CAD Viewer", Page 341

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

- To install or update software version 18, a control with a hard disk size of at least 30 GB is required. The control also needs at least 4 GB RAM.
- The **Test Run** operating mode has been enhanced as follows:
  - In the **Test Run** operating mode, the control uses the active preset.
  - The **RESET THE PRESET** soft key has been added to the **BLANK IN WORK SPACE** menu. For the simulation, you can use this soft key to set the values of the active preset in the principal axis to 0.
- The **OPEN COMPENS. TABLES** soft key has been added to the **Program run, single block** and **Program run, full sequence** operating modes. This soft key allows you to open and edit the active datum table and the active compensation tables.
- In the **Program run, single block** and **Program run, full sequence** operating modes, the **ACTUAL POSITON CAPTURE** key allows you to load the current position values of an axis into the datum table.
- The control can execute NC programs with the **SECTION MONITORING** NC function. This NC function may be included in NC programs of the TNC7, but has no function on the TNC 128.
- The control supports USB data media with the NTFS file system.
- The control provides the additional tool **Parole** that allows you to open video files.
- Within the file management, the control hides system files, as well as files and folders whose name begins with a period. If necessary, you can display the files with the **SHOW HIDDEN FILES** soft key.

- The general status display has been enhanced as follows:
  - The control displays a corresponding icon in the general status display when tool radius compensation is active.
  - If a feed-rate limit has been activated with the **F MAX** soft key, the control displays an exclamation mark after the feed-rate value in the general status display.
- The TS 760 input option has been added to the **TYPE** column of the touch probe table.
- You define the shape of the stylus in the **STYLUS** column of the touch probe table. You define an L-shaped stylus with the **L-TYPE** selection.

- The following tool types have been added:
  - **Face milling cutter,MILL_FACE**
  - **Chamfer mill, MILL_CHAMFER**
  - **Side milling cutter, MILL_SIDE**
- The tool table has been enhanced as follows:
  - In the **RCUTS** column of the tool table, you define the front-face cutting width of a tool (e.g., for indexable inserts).
  - You define the usable length of a tool in the **LU** column of the tool table. The usable length limits the plunging depth of the tool in cycles.
  - You define the neck radius of the tool in the **RN** column of the tool table. This allows the control to display the tool correctly in the simulation (e.g., neck of end mills or side milling cutters).
  - You define a radius at the tip of the tool in the **R_TIP** column of the tool table.
  - You define a database ID for the tool in the **DB_ID** column of the tool table. In a tool database for all machines, you can identify tools with unique database IDs (e.g., within a workshop). This allows you to coordinate the tools of multiple machines more easily.
- The **ACTUAL POSITION CAPTURE** soft key allows you to load the actual position of the tool axis into the form view of the tool management as the tool length.
- The **POS. DISP.** soft key enables you to switch the tool table view. The control displays the tool table in combination with the position display or in full screen mode.
- Compensation tables allow you to compensate for tools during program run without having to edit the NC program or the tool tables. The compensation table *.tco is effective in the tool coordinate system as an alternative to the compensation in the tool call.

- The control supports the TS 760 workpiece touch probe.
- A link to the **Firewall settings** HEROS function has been added to the **External access** MOD function.
- A link to the **Certificate and keys** HEROS function has been added to the **External access** MOD function. This function can be used to define settings for secure connections via SSH.
- If the machine manufacturer has defined the parameter **CfgOemInfo** (no. 131700), then the control displays the **Info about machine manufacturer** area in the **General Information** MOD group.
- The HEROS menu has been enhanced as follows:
  - In the HEROS settings, you can adjust the screen brightness of the control.
  - In the **Screenshot settings** window you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).

- The user administration has been enhanced as follows:
  - When user administration is active, the file manager displays the **public** directory that can be accessed by every user.

    When you place the cursor on the **public** directory, the control shows the **ADVANCED ACCESS RIGHTS** soft key. This soft key allows the owner of a file to define the access rights of the following users:
    - Owner
    - Group
    - Other users
  - The **useradmin**, **oem**, and **sys** users can deactivate the user administration.
  - When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user authentication. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.
  - When user administration is active, you can create user-specific private network drive connections. **Single Sign On** allows you to connect to an encrypted network drive while logging on to the control.
  - When configuring the user administration, you can use the **Autologin** function to define a user who will automatically be logged on by the control during booting.
- The optional machine parameter **applyCfgLanguage** (no. 101305) allows you to define whether the HEROS operating system adopts the conversational language defined in machine parameter **ncLanguage** (no. 101301) during booting. If you activate this function, you can change the conversational language only in the machine parameters.
- The optional machine parameter **extendedDiagnosis** (no. 124204) allows you to define whether the control saves graphics journal data after a restart. This data is used to diagnose graphics problems.
- The machine parameter **CfgTTRectStylus** (no. 114300) has been added. This parameter allows you to define settings for a tool touch probe with a cuboid probe contact.

Modified functions

- To make the control represent the workpiece blank in the simulation, the workpiece blank must have minimum dimensions. The minimum dimensions are 0.1 mm or 0.004 inches in all axes and for the radius.

  Further information: "Defining the workpiece blank: BLK FORM", Page 83
- The pop-up window for tool selection always shows the content of the **NAME** column, even if you are calling the tool with the tool number.

  Further information: "Calling the tool data", Page 121
- Within **FUNCTION S-PULSE**, you can define a lower and an upper speed limit for the pulsing speed by means of the **FROM-SPEED** and **TO-SPEED** syntax elements.

  Further information: "Pulsing spindle speed FUNCTION S-PULSE", Page 302
- In the **TABDATA WRITE**, **TABDATA ADD** and **FN 27: TABWRITE** (ISO: **D27**) NC functions, you can enter values directly.

  Further information: "Accessing table values ", Page 332

  Further information: "FN 27: TABWRITE writing to a freely definable table", Page 298
- If you program a precision stop of rotary axes using **M134** or **M135**, the control no longer displays an error message. The control ignores these miscellaneous functions.
- The number range for the machine manufacturer's miscellaneous functions has been increased from 1999 to 9999.
- The **FN 10** function also allows you to check QS parameters and texts for inequalities.

  Further information: "Programming if-then decisions", Page 217
- You can use UTF-8 character encoding in the mask file of **FN 16: F-PRINT**.

  Further information: "FN 16: F-PRINT – Formatted output of text and Q parameter values", Page 233
- The priority of arithmetic operations has been changed in the Q parameter formula.

  Further information: "Calculation rules", Page 218
- You can use combined QS parameters in the **SQL EXECUTE** and **SQL SELECT** functions.

  Further information: "Accessing tables with SQL statements", Page 264

- While program run is interrupted or has been canceled, you can edit Q and QS parameters with numbers 0 to 99, 200 to 1199 and 1400 to 1999 in the **Q parameter list** window.
- Scrolling in the structure window works in the same way as scrolling in the NC program. You can define the position of the active structure block by soft key.

  Further information: "Structuring NC programs", Page 141
- The control uses the active unit of measure (mm or inches) for calculations in the cutting data calculator.
- The result fields and the diameter field of the cutting data calculator are freely editable.

  Further information: "Cutting data calculator", Page 146
- The CAD Viewer has been enhanced as follows:
  - **CAD Viewer** performs all internal calculations in mm. If you select the inch unit of measure, the **CAD Viewer** converts all values to inches.
  - The **Show sidebar** icon enlarges the Sidebar window to half the size of the screen.
  - The control always shows the **X**, **Y** and **Z** coordinates in the Element Information window. If the 2D mode is active, the control shows the Z coordinate dimmed.
  - **CAD Viewer** also recognizes circles as machining positions that consist of two semicircles.
  - You can save the workpiece preset and workpiece datum information to a file or to the clipboard, even when the software option CAD Import is not available.

  Further information: "CAD Viewer", Page 341
- In the compensation tables (*.tco and *.wco), the input range for all columns containing numerical values has been changed from +/– 999.999 to +/– 999.9999.

  Further information: "Compensation table", Page 328
- The **FILTER** soft key in the error window was renamed to **GROUPING**. With this soft key, the control groups warnings and error messages.

  Further information: "GROUPING soft key", Page 155

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

- If you set the **MEASURING** soft key to **ON**, then the control displays the following additional information:
  - Surface orientation of the current position
  - Workpiece number
  - Workpiece name
  - Note during machining at rapid traverse, thread cutting cycle, or blank form update
- The **BLANK IN WORK SPACE** menu provides a soft key that allows you to load the current machine status. The control loads the following information in addition to the active preset:
  - Active kinematics
  - Active traverse ranges
  - Active machining mode
  - Active traverse limits
- The control uses hatch marking to depict threads in the simulation.
- The simulation takes into account the following columns of the tool table:
  - **R_TIP**
  - **LU**
  - **RN**
- The control takes into account the following NC functions in the **Test Run** operating mode.
  - **FN 27: TABWRITE** (ISO: **D27**)
  - **FUNCTION FILE**
  - **FUNCTION FEED DWELL**
- Any display filter you have set in the file manager will be retained even after a control restart.
- If you create a table, and there is at least one prototype available for this file type, the control displays the window **Select table format**. The control also shows whether the prototype is defined in mm or inches. If the control shows both units of measure, you can select a unit of measure.

  The machine manufacturer defines the prototypes. If the prototype contains values, the control transfers these values to the newly created table.

- If you exit an NC program by pressing the **END** key, the control opens the file manager. The cursor is on the NC program that was just closed. If you press the **END** key again, the control opens the original NC program again with the cursor on the last selected line. With large files this behavior can cause a delay.
- The machine manufacturer defines the sequence for traversing the axes when returning to the contour.
- The control takes into account manual axes when returning to the contour.
- In **Program run, single block** operating mode, the control interprets the workpiece blank definition as a single NC block only.
- The control displays the tool index in the block scan pop-up window if needed.
- The control considers the functions **FN 27: TABWRITE** (ISO: D27) and **FUNCTION FILE** only in the operating modes **Program run, single block** and **Program run, full sequence**.
- The additional status display has been enhanced as follows:
  - The control also displays the number of repetitions on the **Overview** and **LBL** tabs of the additional status display after an internal stop.
  - On the **TT** tab of the additional status display, the control displays the tilt angle of the tool touch probe as well as information about the cuboid probe contacts.
  - In **Test Run** operating mode, the control displays the **M** tab of the additional status display when the screen layout **PROGRAM STATUS** is active.
- The handwheel functions have been enhanced as follows:
  - The smallest definable speed level on handwheels with display has been changed from 0.1% to 0.01% of the maximum handwheel speed.
  - If a handwheel is active, the control shows the contouring feed rate in the display during program run. If only the currently selected axis is moving, the control shows the axis feed rate.
  - When you activate a handwheel with display, the control automatically activates the override potentiometer of the handwheel.
  - In the operating modes **Manual Operation** and **Positioning w/ Manual Data Input**, you can activate a handwheel with display while a macro or a manual tool change is being executed.
- You can activate or deactivate the **F MAX** soft key for reducing the feed rate. The defined value is retained.
- The minimum input value of the **FMAX** column in the touch probe table has been changed from –9999 to +10.
- The form view of the tool management shows only those input fields that are needed for the selected tool type.
- The maximum input range of the **LTOL** and **RTOL** columns of the tool table has been increased. It was from 0 mm to 0.9999 mm, and is now from 0.0000 mm to 5.0000 mm.
- The maximum input range of the **LBREAK** and **RBREAK** columns of the tool table has been increased. It was from 0 mm to 0.9999 mm, and is now from 0.0000 mm to 9.0000 mm.

- The control no longer supports the ITC 750 additional operating station.
- If the control is accessed from external, it displays a corresponding icon in the header.

  The control uses an icon to indicate whether a connection configuration is secure or non-secure.
- Limits that have been defined in the **Traverse limits** MOD function are also effective for modulo axes.
- **Program run** in the **Machine times** MOD area shows only the times at which at least one axis was moving during program run.
- From within the **Diagnostic functions** MOD group, you can access **TNCdiag** and **Hardware configuration** without a code number.
- The user interface of the **Network settings** window has been changed. For network configuration, use the **Network Connections** window.
- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.
- You can export and import existing network configurations in the **Network settings** window.

- If you enter a password or code number with Caps Lock active, then the control issues a message.
- The machine manufacturer can define a path for saving the values of the QR parameters. If the values are on the **TNC** drive, you can use the **NC/PLC Backup** HEROS function to back up the QR parameters.
- **PKI Admin** now includes the **Advanced settings** tab.

  You can define whether the server certificate should contain static IP addresses and allow connections without an associated CRL file.
- The user administration has been enhanced as follows:
  - When user administration is active, the **Liberating motion** mode requires the NC.OPModeManual permission (i.e., at least the role of **NC.Programmer**).
  - If you use the **Connection to Windows domain** function when configuring the user administration, you can set up a secure connection by activating the **Use LDAPs** check box.
  - If a remote log-in takes place while user administration is inactive, for example via SSH, the control automatically assigns the **HEROS.LegacyUserNoCtrlfct** role.
  - If you deactivate the user administration and select the **Delete existing user databases** check box, the control also deletes the .home folder in the **TNC:** directory.
  - Your IT administrator can set up a function user to facilitate connectivity to the Windows domain.
  - If you have connected the control to the Windows domain, you can export the required configurations for other controls.
- The machine parameter **spindleDisplay** (no. 100807) has been enhanced. The control can also display the spindle position on the **Overview** tab of the additional status display when the spindle is in jog mode.
- The input range of machine parameter **displayPace** (no. 101000) has been extended. The minimum display step of the axes is 0.000001° or mm.
- If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

  When the control detects a non-secure connection, it displays an informational notice.
- Machine parameter **CfgStretchFilter** (no. 201100) has been removed.

Modified cycle functions with 77184x-18

> **Overview of new and modified software functions**
>
> Further information about the previous software versions is presented in the **Overview of New and Modified Software Functions** documentation. Please contact HEIDENHAIN if you need this documentation.
>
> ID: 1322088-xx

- In Cycle **12 PGM CALL** (ISO: G39) you can use the **SYNTAX** soft key to place paths within quotation marks. To separate folders and files within paths, both the **** and the **/** character are permitted.

  Further information: "Cycle 12 PGM CALL ", Page 484
- Cycles **202 BORING** (ISO: **G202**) and **204 BACK BORING** (ISO: **G204**) restore the spindle status after machining to that which was active before the cycle.

  Further information: "Cycle 202 REAMING ", Page 391

  Further information: "Cycle 204 BACK BORING ", Page 401
- The parameter **Q373 FEED AFTER REMOVAL** has been added to Cycle **205 UNIVERSAL PECKING** (ISO: **G205**). This parameter is used to define the feed rate for returning to the advanced stop distance after chip removal.

  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 405
- Cycles **205 UNIVERSAL PECKING** (ISO: **G205**) and **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**) check the parameter **Q379 STARTING POINT**. If the value of the starting point is equal to or greater than the value of the parameter **Q201 DEPTH**, then the control issues an error message.

  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 405

  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 413
- Parameters **Q429 COOLANT ON** and **Q430 COOLANT OFF** in Cycle **241 SINGLE-LIP D.H.DRLNG** (**G241**) have been extended. You can define a path for a user macro.

  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 413
- Cycle **240 CENTERING** (ISO: **G240**) has been expanded in order to consider pre-drilled diameters.

  The following parameters have been added:
  - **Q342 ROUGHING DIAMETER**
  - **Q253 F PRE-POSITIONING**: If parameter **Q342** is defined, feed rate for approaching the deepened starting point

    Further information: "Cycle 240 CENTERING ", Page 382
- The machine manufacturer can hide the cycles **220 POLAR PATTERN** (ISO: **G220**) and **221 CARTESIAN PATTERN** (ISO: **G221**). We recommend using the **PATTERN DEF** function.

  Further information: "Pattern definition with PATTERN DEF", Page 360
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), you program a limit that is perpendicular to the milling direction **Q350**, then the

control adds the tool radius to the length of the surface in the unlimited direction. As a result, the control completely machines the defined surface without leaving behind any residual material, as would be caused by the tool radius. If the parameter **Q220** (corner radius) is defined, then the control adds both the tool radius and this value to the length of the surface.

Further information: "Cycle 233 FACE MILLING ", Page 456

- If, in Cycle **233 FACE MILLING** (ISO: **G233**), the parameter **Q389** has been defined with the value 2 or 3 and a lateral limit is defined in addition, then the control approaches the contour or departs from it on an arc with **Q207 FEED RATE MILLING**.

  Further information: "Cycle 233 FACE MILLING ", Page 456
- Cycle **253 SLOT MILLING** monitors a cutting width defined in the column **RCUTS** of the tool table. If the center of a tool that is not a center-cut tool would contact the workpiece surface, the control issues an error message.
  Further information: "Cycle 253 SLOT MILLING ", Page 444
- Cycle **251 RECTANGULAR POCKET** takes into consideration a cutting width defined in the column **RCUTS** when calculating the plunging path.

  Further information: "Cycle 251 RECTANGULAR POCKET ", Page 439
- If the defined usable length in column **LU** of the tool table is less than the depth, the control displays an error message.

  The following cycles monitor the usable length LU:
  - All cycles for drilling and boring
  - All cycles for tapping
  - All cycles for the machining of pockets and studs

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**, option 17) allow you to calibrate a tool touch probe with cuboid probe contacts.

  Further information: "Cycle 480 CALIBRATE TT (option 17)", Page 500

  Further information: "Cycle 484 CALIBRATE IR TT (option 17)", Page 502
- The parameter **Q523 TT-POSITION** has been added to Cycle **484 CALIBRATE IR TT** (ISO: **G484**). This parameter allows you to define the position of the tool touch probe and, if desired, to transfer the position to the machine parameter **centerPos** after calibration.

  Further information: "Cycle 484 CALIBRATE IR TT (option 17)", Page 502
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**, option 17) first measures the tool length and then the tool radius.

  Further information: "Cycle 483 MEASURE TOOL (option 17)", Page 512
- Using the optional machine parameter **maxToolLengthTT** (no. 122607), the machine manufacturer defines a maximum tool length for tool touch probe cycles.

  Further information: "Measuring a tool of length 0", Page 495
- Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of

parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

Further information: "Setting machine parameters", Page 496

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:

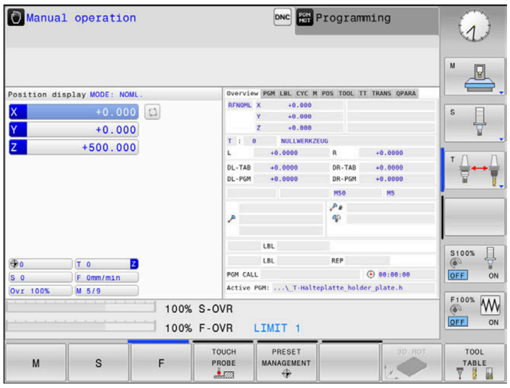
- Switching on the machine
- Programming the workpiece

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: hazard to the user!

Machines and machine components always pose 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.

To switch on the machine:

- Switch on the power supply for the control and the 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

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

2.3 Programming the first part

Selecting the operating mode

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

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

Further information on this topic

- Operating modes
  Further information: "Programming", Page 76

Important controls and displays

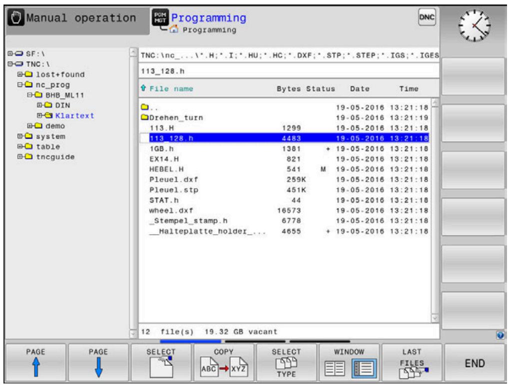
Key	Functions for conversational guidance
ENT	Confirm entry and activate the next dialog prompt
NO ENT	Ignore the dialog question
END	End the dialog immediately
DEL	Abort dialog, discard entries
	Soft keys on the screen with which you select functions appropriate to the active operating state

Further information on this topic

- Writing and editing NC programs
  Further information: "Editing an NC program", Page 90
- Overview of keys
  Further information: "Controls and displays", Page 2

Creating a new NC program / file management

To create a new NC program:



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.
- ▶ Select a folder

GOTO □

- ▶ Press the **GOTO** key
- > The control opens a screen keyboard in a pop-up window.
- ▶ Enter the 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

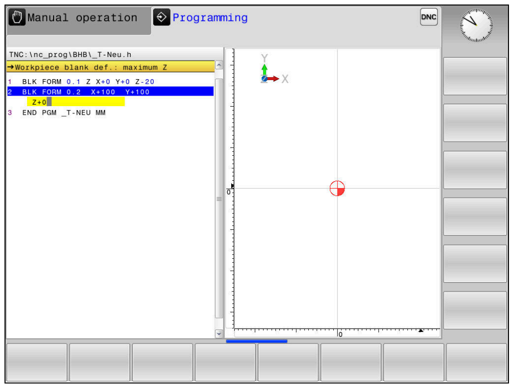
- ▶ Press the soft key of the desired unit of measure: **MM** or **INCH**

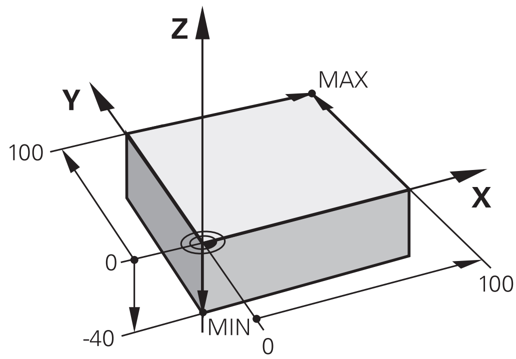
The control automatically generates the first and last NC blocks of the NC program. You will not be able to change these NC blocks at a later time.

Further information on this topic

- File management
  Further information: "File management", Page 96
- Creating a new NC program
  Further information: "Creating and entering NC programs", Page 82

Defining a workpiece blank





Once you have opened a new NC program, you can define a workpiece blank. You can define a cuboid by entering the MIN and MAX points relative to the selected preset.

After you have selected the desired shape for the blank with the appropriate soft key, the control automatically initiates the workpiece blank definition process and prompts you to enter the required data.

To define a cuboid-shaped blank:

- Press the soft key for the desired shape of the workpiece blank
- **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 blank relative to the preset (e.g.: 0), and confirm with the **ENT** key
- **Workpiece blank def.: minimum Y**: Enter the smallest Y coordinate of the blank relative to the preset (e.g., 0), and confirm with the **ENT** key
- **Workpiece blank def.: minimum Z**: Enter the smallest Z coordinate of the blank relative to the preset (e.g., –40), and confirm with the **ENT** key
- **Workpiece blank def.: maximum X**: Enter the largest X coordinate of the blank relative to the preset (e.g., 100), and confirm with the **ENT** key
- **Workpiece blank def.: maximum Y**: Enter the largest Y coordinate of the blank relative to the preset (e.g., 100), and confirm with the **ENT** key
- **Workpiece blank def.: maximum Z**: Enter the largest Z coordinate of the blank relative to the preset (e.g., 0), and confirm with the **ENT** key

> The control ends the dialog.

The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

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 85

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 M3
5 X...   R0 FMAX
6 Z+10 R0  F3000 M8
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; turn on spindle
3 Pre-position the tool in the working plane near the contour starting point
4 Pre-position the tool along the tool axis above the workpiece, or pre-position the tool directly to the cutting depth, and turn on coolant as needed
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 128

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 M3
5 PATTERN DEF  POS1( X...  Y... Z...  ) ...
6 CYCL DEF...
7 CYCL CALL PAT  FMAX M8
8 Z+250 R0  FMAX M2
9 END PGM  BSBCYC MM
```

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

Further information on this topic

- Cycle programming
  Further information: "Fundamentals / Overviews", Page 345

Programming a simple contour

Suppose you want to mill a single time around the contour shown on the right at a depth of 5 mm. You have already defined the workpiece blank.

After you have opened an NC block with a function key, the control will prompt you to enter all of the data in the header using dialog texts.

To program the contour:

Call the tool

TOOL CALL

- Press the **TOOL CALL** key
- Enter the tool data, e.g., tool number 16

ENT

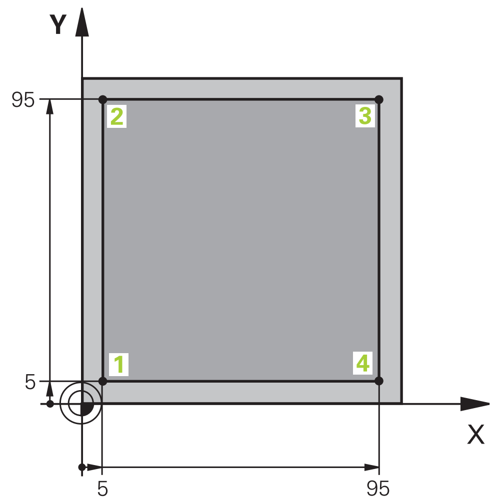
- Press the **ENT** key

ENT

- Confirm the tool axis **Z** with the **ENT** key
- Enter the spindle speed (e.g., 6500)

END

- Press the **END** key
- > The control completes the NC block.



The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

Retract the tool

Z

- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**, which means there is no radius compensation.

ENT

- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**, such as **M3**, turn on spindle

END

- Press the **END** key
- > The control saves the positioning block.

Pre-position the tool in the working plane

X

- Press the **X** axis key
- Enter the value for the position to be approached (e.g., –20 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**.

ENT

- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**

END

- Press the **END** key
- > The control saves the positioning block.

Y

- Press the **Y** axis key
- Enter the value for the position to be approached (e.g., –20 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**.

ENT

- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**

END

- Press the **END** key
- > The control saves the positioning block.

Pre-positioning the tool to the cutting depth

Z

- Press the **Z** axis key
- Enter the value for the position to be approached (e.g., -5 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**.
- Enter the value for the positioning feed rate (e.g., 3000 mm/min)

ENT

- Press the **ENT** key
- Enter a miscellaneous function **M**, such as **M8** to turn coolant on

END

- Press the **END** key
- > The control saves the positioning block.

Machine the contour

X

- Press the **X** axis key
- Enter the X coordinate of contour point **1** (e.g., **X 5**)

ENT

- Press the **ENT** key

R -

- Press the **R-** soft key
- > The control shortens the traverse path by an amount equivalent to the tool radius.
- Enter the value for the positioning feed rate (e.g., 700 mm/min)

ENT

- Press the **ENT** key
- If needed, enter a miscellaneous function **M**

END

- Press the **END** key
- > The control saves the positioning block.

Y

- Press the **Y** axis key
- Enter the changing coordinate of contour point **2** (e.g., **Y 95**)

ENT

- Press the **ENT** key

R+

- Press the **R+** soft key

END

- Press the **END** key
- > The control applies the changed value and retains all of the other information from the previous NC block.

X

- Press the **X** axis key
- Enter the changing coordinate of contour point **3** (e.g., **X 95**)

ENT

- Press the **ENT** key

R+

- Press the **R+** soft key

END

- Press the **END** key

Y

- Press the **Y** axis key
- Enter the changing coordinate of contour point **4** (e.g., **Y 5**)

ENT

- Press the **ENT** key

R+

- Press the **R+** soft key

END

- Press the **END** key

Complete the contour and depart from it

X
- Press the **X** axis key
- Enter the X coordinate of contour point **1**

ENT
- Press the **ENT** key

R+
- Press the **R+** soft key

END
- Press the **END** key

X
- Press the **X** axis key
- Enter the value for the position to be approached (e.g., –20 mm)

ENT
- Press the **ENT** key

ENT
- At radius compensation, press **ENT**
- > The control applies **R0**.
- Enter the value for the positioning feed rate (e.g., 3000 mm/min)

ENT
- Press the **ENT** key
- Enter a miscellaneous function **M**, such as **M9** to turn off the coolant

END
- Press the **END** key
- > The control saves the departure movement.

Retract the tool

Z

- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT

- Press the **ENT** key

ENT

- At radius compensation, press **ENT**
- > The control applies **R0**, which means there is no radius compensation.

ENT

- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**, such as **M30**, program end

END

- Press the **END** key
- > The control saves the positioning block and ends the NC program.

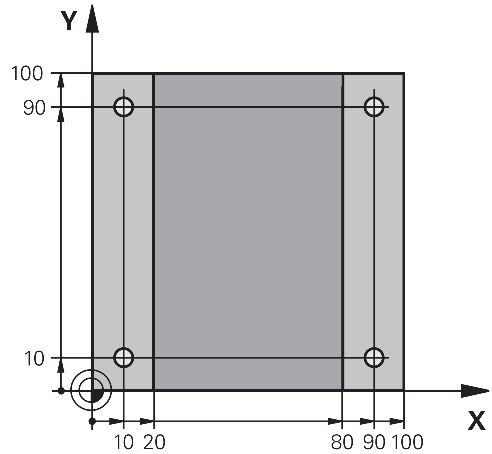
Further information on this topic

- Creating a new NC program
  Further information: "Creating and entering NC programs", Page 82
- Programmable feed rates
  Further information: "Possible feed rate input", Page 88
- Tool radius compensation
  Further information: "Tool radius compensation", Page 125
- Miscellaneous functions M
  Further information: "Miscellaneous functions for program run inspection, spindle and coolant ", Page 169

Creating a cycle program

Suppose that you are tasked with drilling the holes shown to the right with a standard drilling cycle (depth: 20 mm). You have already defined the workpiece blank.

Call the tool



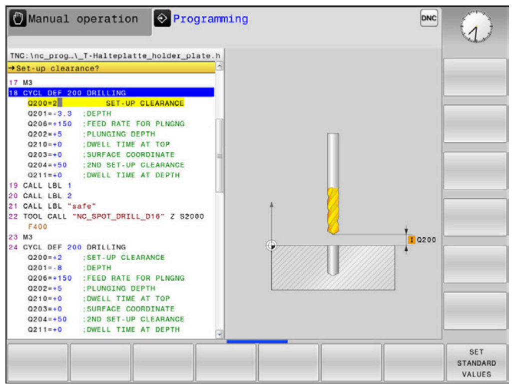
TOOL CALL
- Press the **TOOL CALL** key
- Enter the tool data, e.g., tool number 5

ENT
- Press the **ENT** key

ENT
- Confirm the tool axis **Z** with the **ENT** key
- Enter the spindle speed (e.g., 4500)

END
- Press the **END** key
- > The control completes the NC block.

Retract the tool



Z
- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT
- Press the **ENT** key

ENT
- At radius compensation, press **ENT**
- > The control applies **R0**, which means there is no radius compensation.

ENT
- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**, such as **M3**, turn on spindle

END
- Press the **END** key
- > The control saves the positioning block.

Define a pattern

- Press the **SPEC FCT** key
- > The control opens the soft key row containing the special functions.
- Press the **CONTOUR + POINT MACHINING** soft key
- Press the **PATTERN DEF** soft key
- Press the **POINT** soft key
- Enter the coordinates of the first position
- Confirm each entry with the **ENT** key
- Press the **ENT** key
- > The control opens the dialog for the next position.
- Enter the coordinates
- Confirm each entry with the **ENT** key
- Enter the coordinates of all positions
- Press the **END** key
- > The control saves the NC block.

Define the cycle

- Press the **CYCL DEF** key
- Press the **DRILLING/ THREAD** soft key
- Press the **200** soft key
- > The control starts the dialog for cycle definition.
- Enter the cycle parameters
- Confirm each entry with the **ENT** key
- > The control displays a graphic illustrating the respective cycle parameter.

Call the cycle

- Press the **CYCL CALL** key
- Press the **CYCLE CALL PAT** soft key
- Press the **ENT** key
- > The control applies **FMAX**.
- If needed, enter a miscellaneous function **M**
- Press the **END** key
- > The control saves the NC block.

Retract the tool

Z
- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT
- Press the **ENT** key

ENT
- At radius compensation, press **ENT**
- > The control applies **R0**.

ENT
- At feed rate **F**, press the **ENT** key
- > The control applies **FMAX**.
- Enter a miscellaneous function **M**, such as **M30** for program end

END
- Press the **END** key
- > The control saves the positioning block and ends the NC program.

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 M3	Retract the tool; turn on spindle
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 M8	Turn on coolant; call cycle
8 Z+250 R0 FMAX M30	Retract the tool, end program
9 END PGM C200 MM	

Further information on this topic

- Creating a new NC program
  Further information: "Creating and entering NC programs", Page 82
- Cycle programming
  Further information: "Fundamentals / Overviews", Page 345

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.

The 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

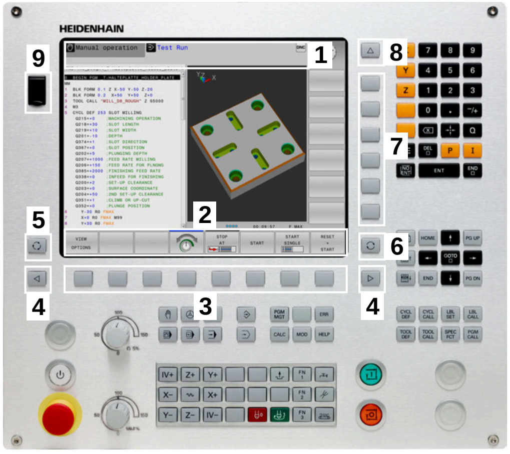
NC programs created on the HEIDENHAIN TNC 124 straight-cut control may not always run on the TNC 128. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.

3.2 Visual display unit and operating panel

Display screen

The control is shipped with a 12.1-inch screen.

The figure at right shows the keys and controls on the VDU:



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 manufacturer's

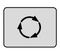
8 Keys for switching the soft keys for machine manufacturer's

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 up the screen layout:



- Press the **screen layout** key: The soft-key row shows the available layout options
  Further information: "Modes of operation", Page 76

PROGRAM + GRAPHICS

- Select the desired screen layout with a soft key

Operating panel

The TNC 128 can be delivered with an integrated operating panel.

1 Machine operating panel
  Further information: Machine manual
2
  - File manager
  - 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 manufacturers do not use the standard HEIDENHAIN operating panel.
External keys e.g., **NC START** or **NC STOP**, are described in your machine manual.

Cleaning

Switch the control off before cleaning the keyboard unit.

NOTICE

Caution: danger of property damage

Incorrect cleaning agents and incorrect cleaning procedures can damage the keyboard unit or parts of it.

- Use permitted cleaning agents only
- Use a clean, lint-free cleaning cloth to apply the cleaning agent

The following cleaning agents are permitted for the keyboard unit:

- Cleaning agents containing anionic surfactants
- Cleaning agents containing nonionic surfactants

The following cleaning agents are prohibited for the keyboard unit:

- Cleaning agents for machines
- Acetone
- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

> Wear operating gloves to prevent the keyboard unit from getting dirty.

If a trackball is embedded in the keyboard, you need to clean it only when it stops functioning properly.

To clean a trackball (if needed):

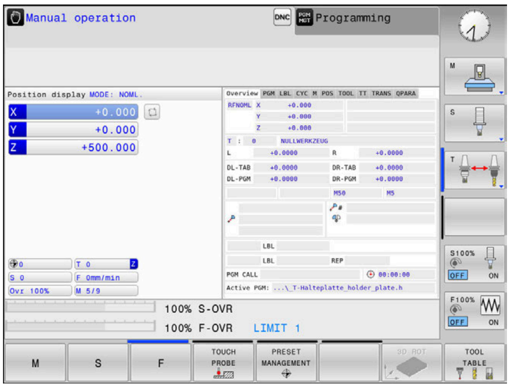
- Switch off the control
- Turn the pull-off ring by 100° in counterclockwise direction
- > Turning the removable pull-off ring moves it upwards out of the keyboard unit.
- Remove the pull-off ring
- Take out the ball
- Carefully remove sand, chips, or dust from the shell area

> Scratches in the shell area may impair the functionality or prevent proper functioning.

- Apply a small amount of the cleaning agent onto a cleaning cloth
- Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

3.3 Modes of operation

Manual Operation and El. Handwheel



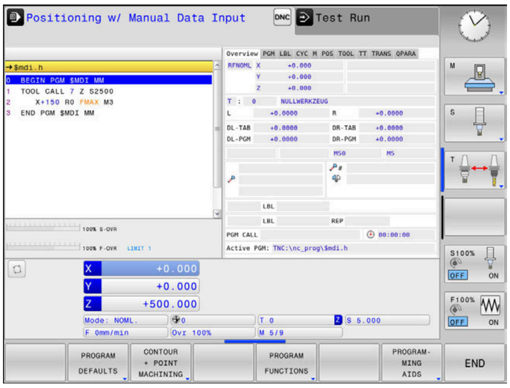
In the **Manual operation** mode of operation, you can set up the machine. You can position the machine axes manually or incrementally, and you can set presets.

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

Soft keys for selecting the screen layout

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

Positioning with Manual Data Input

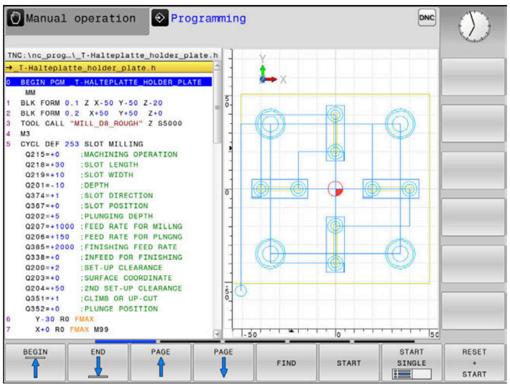


This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

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



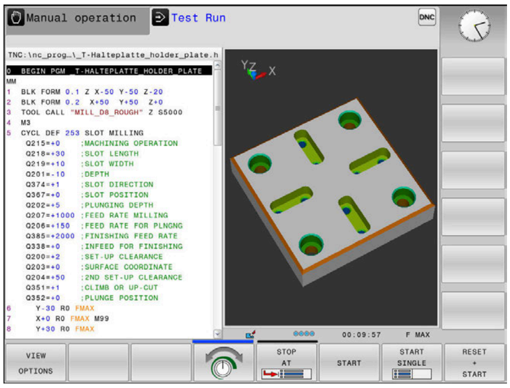
In this mode of operation you create NC programs. The various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

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 simulates NC programs and program sections in order to check them 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.



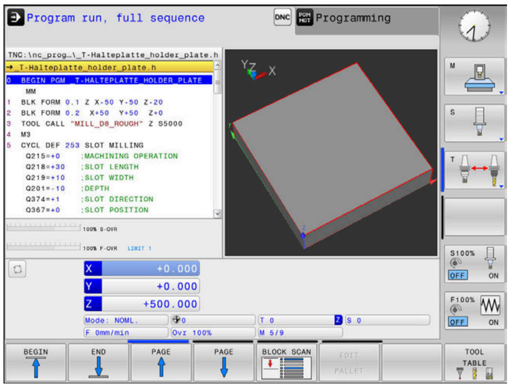
Soft keys for selecting the screen layout

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. The workpiece blank definition will be interpreted as a separate NC block.

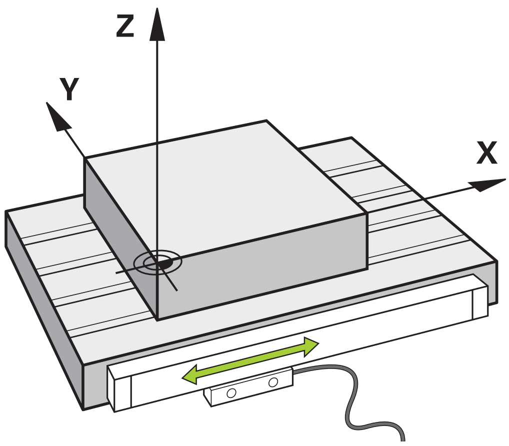


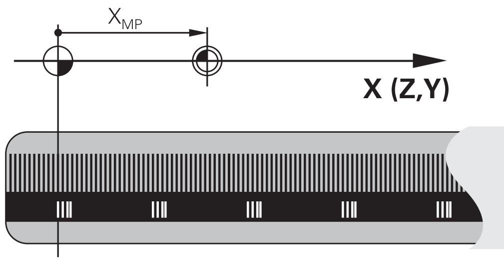
Soft keys for selecting the screen layout

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





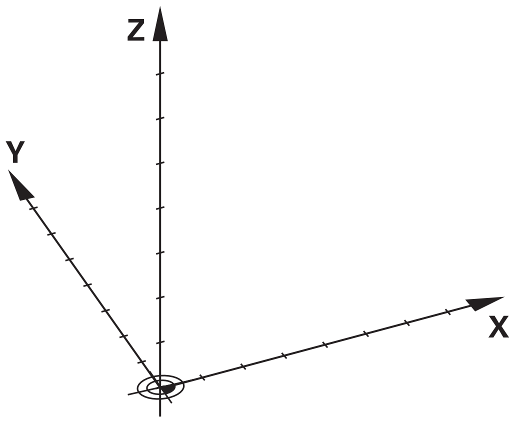
The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders.

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

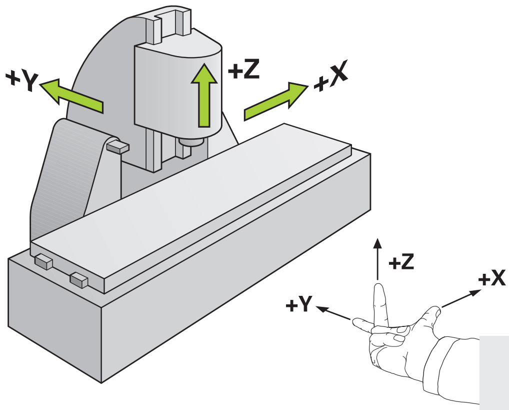


A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

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

The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

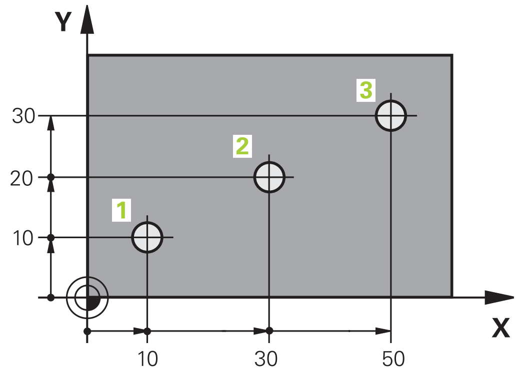
Absolute and incremental workpiece positions

Absolute workpiece positions

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

Example 1: Holes dimensioned in absolute coordinates

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

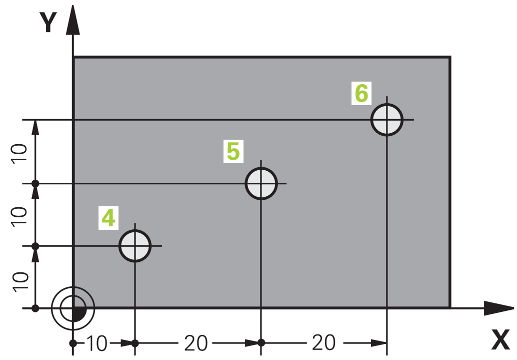


Incremental workpiece positions

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

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

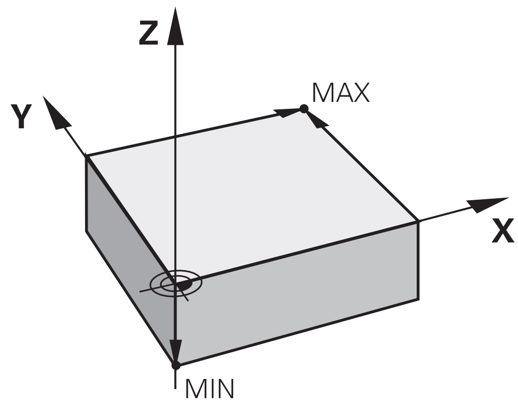
Example 2: Holes dimensioned in incremental coordinates



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

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

Selecting the preset



A production drawing specifies a certain form element of the workpiece (usually a corner) as the absolute reference point (datum). When setting the preset, first align the workpiece along the machine axes, and move the tool to a known position in each axis relative to the workpiece. For each position, set the display of the control either to zero or to a known position value. You thereby assign the workpiece to the reference system that is applicable for the control's display or your NC program.

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

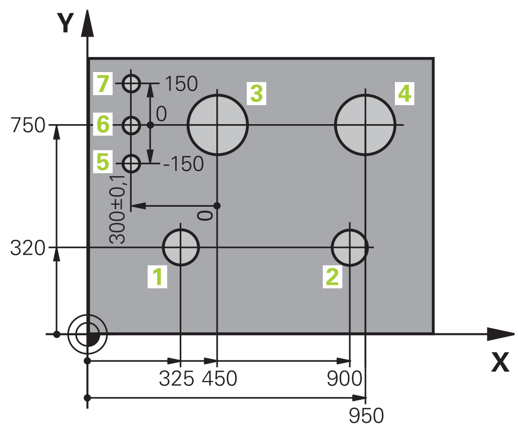
Further information: "Cycle 7 DATUM SHIFT ", Page 471

If the production drawing is not dimensioned for NC programming, then select a position or corner of the workpiece as a reference point from which the dimensions of the remaining workpiece positions can be determined.

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 relative to an absolute preset with the coordinates X=0 Y=0. The coordinates of holes **5** to **7** refer to the relative preset with the absolute coordinates X=450 Y=750. A **Datum shift** allows you to temporarily shift the datum to the position X = 450, Y = 750 in order for you to program the holes (**5** to **7**) without further calculations.



3.5 Creating and entering NC programs

Structure of an NC program in HEIDENHAIN Klartext

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

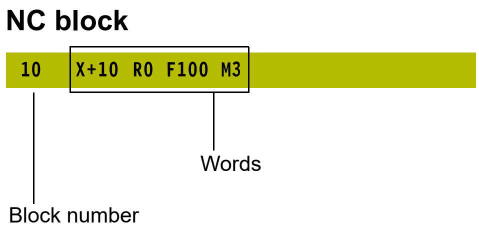
The control numbers the NC blocks of an NC program in ascending sequence.

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.



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 workpiece 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, then press the **SPEC FCT** key, the **PROGRAM DEFAULTS** soft key, and then the **BLK FORM** soft key. The control needs this definition for its graphical simulations.

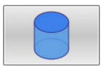
- You only need to define the workpiece blank if you wish to run a graphic test for the NC program.
- To make the control represent the workpiece blank in the simulation, the workpiece blank must have minimum dimensions. The minimum dimensions are 0.1 mm or 0.004 inches in all axes and for the radius.
- The **Advanced checks** function in the simulation uses the information from the workpiece blank definition for workpiece monitoring. Even if several workpieces are clamped in the machine, the control can monitor only the active workpiece blank!

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

The control’s full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

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

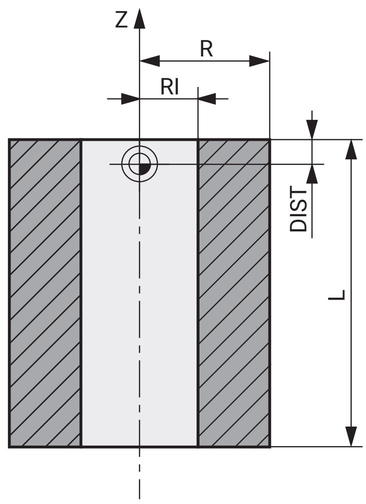
Example

0 BEGIN PGM NEW MM	Program beginning, 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



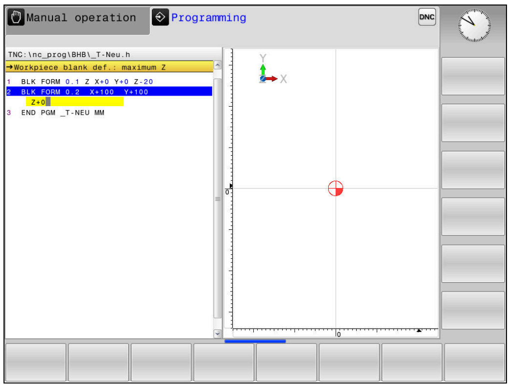
> The parameters **DIST** and **RI** or **DI** are optional and need not be programmed.

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

An NC program is always entered in **Programming** mode. Example for creating a program:



- 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**

> The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
>
> Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

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

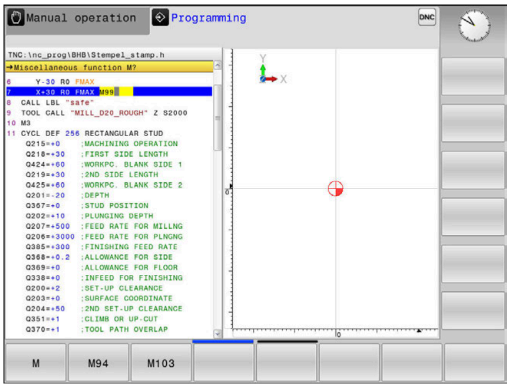
0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Spindle axis, MIN point coordinates
2 BLK FORM 0.2 X+100 Y+100 Z+0	MAX point coordinates
3 END PGM NEW MM	Program end, name, unit of measure

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

> If you do not wish to define a workpiece blank, then 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:

- ▶ 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 an 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
PAGE ↑	Go to previous page
PAGE ↓	Go to next page
BEGIN ↑	Go to beginning of program
END ↓	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
GOTO □	Select a specific NC block **Further information:** "Using the GOTO key", Page 134

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
- Initiate the dialog

Save changes

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

- 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 abort the process by pressing the **CANCEL** soft key

> The file saved with **SAVE AS** can also be found in the file manager 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

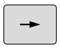
- ▶ Press the **CANCEL CHANGE** soft key
- > The control opens a window in which you can confirm or cancel this action.
- ▶ Confirm with the **YES** soft key or cancel with the **ENT** key, or press the **NO** soft key to abort

Editing and inserting words

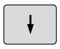
- ▶ Select a word in an NC block
- ▶ Overwrite it with the new value
- > The dialog is available while the word is highlighted.
- ▶ To accept the change, press the **END** key

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



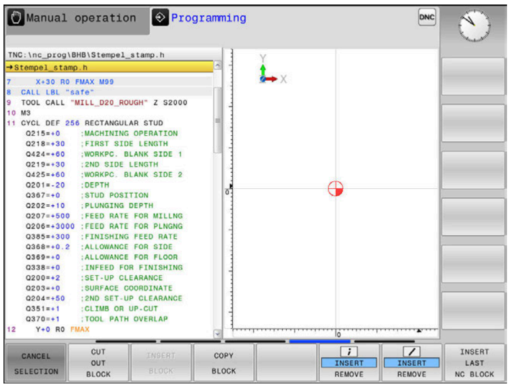
- ▶ Select an NC block with the arrow keys
  - Arrow down: search forwards
  - Arrow up: search backwards

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:

- ▶ 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.

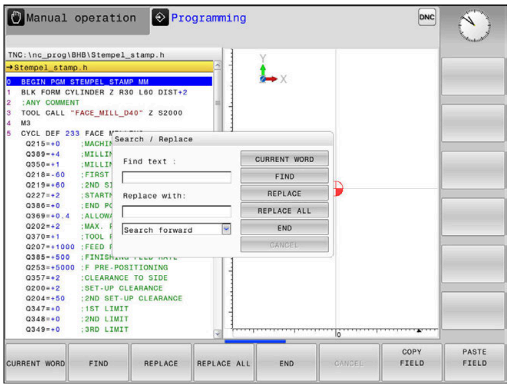
> If you want to transfer a program section to another NC program, you now need to select the desired NC program in the file manager.

- ▶ 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
- ▶ 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 of the found syntax elements without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a result, NC programs may be irreversibly damaged.

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

The **FIND** and **REPLACE** functions cannot be used in the active NC program while the program is running. These 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. Or, to replace all text occurrences, press the **REPLACE ALL** soft key. Or, 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 file name extension *.bak after editing and saving 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.

> 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 data are input or read.

> 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 98

Displaying externally generated files on the control

The control features several software 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
Graphic 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 ****.

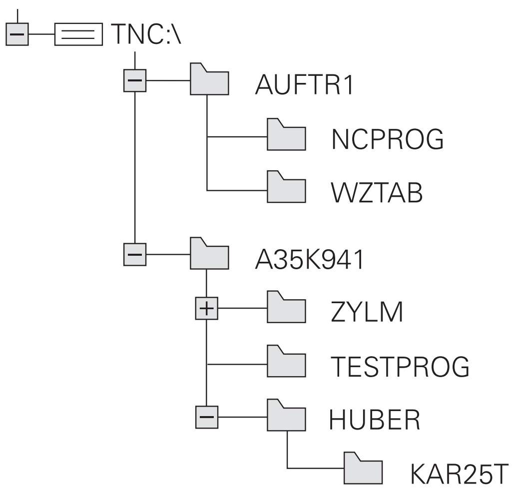
The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

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.



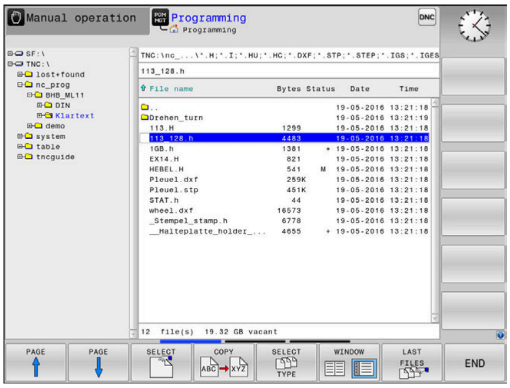
Overview: Functions of the file manager

Soft key	Function	Page
COPY ABC → XYZ	Copy a single file	103
SELECT TYPE	Display a specific file type	101
NEW FILE	Create new file	103
LAST FILES	Display the last 10 files that were selected	106
DELETE	Delete a file	107
TAG	Tag a file	108
RENAME ABC = XYZ	Rename file	109
PROTECT	Protect a file against editing and erasure	110
UNPROTECT	Cancel file protection	110
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	301
NET	Manage network drives	See the User's Manual for Setup, Testing and Running NC Programs
SELECT EDITOR	Select the editor	110
SORT	Sort files by properties	109
COPY DIR	Copy a directory	106
DELETE ALL	Delete directory with all its subdirectories	
UPDATE TREE	Refresh directory	
RENAME ABC = XYZ	Rename a directory	
NEW DIRECTORY	Create a new directory	

Calling the File Manager

PGM MGT

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



> If you exit an NC program by pressing the **END** key, the control opens the file manager. The cursor is on the NC program that was just closed.
>
> If you press the **END** key again, the control opens the original NC program again with the cursor on the last selected line. With large files this behavior can cause a delay.
>
> If you press the **ENT** key, the control always opens an NC program with the cursor on line 0.

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	The file has been selected in the **Programming** mode of operation
S	File is selected in the **Test Run** operating mode
M	The file is selected in a Program Run mode of operation
+	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 deletion and editing, because it is being run
Date	Date that the file was last edited
Time	Time that the file was last edited

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

Selecting drives, directories and files

PGM MGT

- Call the file manager by pressing the **PGM MGT** key

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

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

PAGE

PAGE

- Moves the cursor one page up or down within a window

Step 1: Select drive

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

SELECT

- Select a drive: Press the **SELECT** soft key, or

ENT

- Press the **ENT** key

Step 2: Select a directory

- Move the highlight to the desired directory in the left 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 **SHOW ALL** soft key
- 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.

Filtering the display

To filter the displayed files:

- Press the **SELECT TYPE** soft key
- Press the soft key for the desired file type

Alternative:

- Press the **SHOW ALL** soft key
- > The control displays all files in this folder.

Alternative:

- Use wildcards, such as **4*.H**
- > The control will show all files of file type .h whose name starts with 4.

Alternative:

- Enter the file name extension, e.g. ***.H;*.D**
- > The control will show all files of file type .h and .d.

Any display filter you have set will remain effective even after a control restart,

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

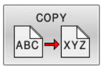
- Select the directory in the left window in which you wish to create the new file
- Position the cursor in the right window

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

- Press the **ENT** key
- > As needed, the control continues the dialog (e.g., select unit of measure).
- Continue the dialog where necessary

Copying a single file

- Move the cursor to the file you wish to copy



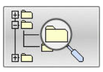
- Press the **COPY** soft key to select the copying function
- > The control opens a pop-up window.

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

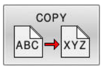
- Press the **SHOW TREE** soft key
- Move the cursor to the directory into which you wish to copy the files,

In the left window

- Press the **SHOW TREE** soft key
- Select the directory with the files that you want to copy, and display the files with the **SHOW FILES** soft key

- 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 108

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!

The **REPLACE FIELDS** function overwrites all lines of the target file that are contained in the copied table 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 accordingly 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.

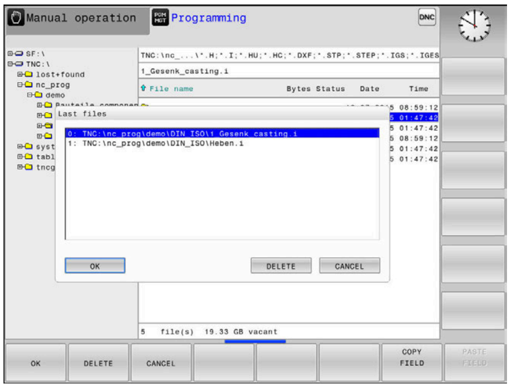
Proceed as follows:

- ▶ 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

- 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 control does not perform an automatic backup of the file prior to deletion (e.g., there is no recycle bin). Files are thereby irreversibly deleted.

- 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 control does not perform an automatic backup of the files prior to deletion (e.g., there is no recycle bin). Files are thereby irreversibly deleted.

- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the directory you want to delete

- Press the **DELETE ALL** 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:

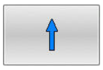
- Move the cursor to the first file

TAG

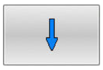
- To display the tagging functions, press the **TAG** soft key

TAG FILE

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



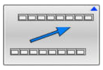
- Move the cursor to other files



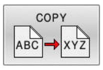
TAG FILE

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

To copy tagged files:

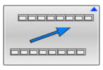


- Leave the active soft-key row



- Press the **COPY** soft key

To delete tagged files:

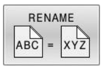


- Leave the active soft-key row

- Press the **DELETE** soft key

Renaming a file

- Move the cursor to the file you wish to rename



- 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 miscellaneous 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**)
- 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:

- 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**

ADVANCED ACCESS RIGHTS

The **ADVANCED ACCESS RIGHTS** function can only be used in connection with user administration. This function requires the **public** directory.

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

Upon the first activation of user administration, the **public** directory below the **TNC:** drive will be connected.

> Access rights can only be defined for files located in the **public** directory.
>
> For all files stored on the **TNC:** drive instead of the **public** directory, the **user** function user will automatically be assigned as the owner.

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

Displaying hidden files

The control hides system files, as well as files and folders whose name begins with a period.

> **NOTICE**
>
> **Caution: Possible loss of data!**
>
> The control's operating system uses certain hidden folders and files. These folders and files are hidden by default. Any manipulation of the system data within the hidden folders might damage the control's software. If you save your own files to these folders, the system will create invalid paths.
>
> - Always leave hidden folders and files hidden
> - Do not use hidden folders and files for saving your own data

If required, you can show the hidden files and folders temporarily, e.g., if a file whose name begins with a period is transferred inadvertently.

To show hidden files and folders:

- Press the **MORE FUNCTIONS** soft key

- Press the **SHOW HIDDEN FILES** soft key
- > The control displays the files and folders.

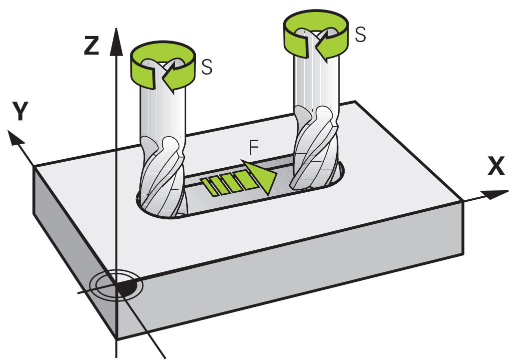
4

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 **FMAX.** 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.

> Make sure to program rapid traverse movements exclusively with the **FMAX** NC function instead of entering extremely high numerical values. This is the only way to ensure that rapid traverse is active on a block-by-block basis and that you can control rapid traverse independently of the machining 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 **FMAX** 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 only reduces the programmed feed rate, and 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, tool number, 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

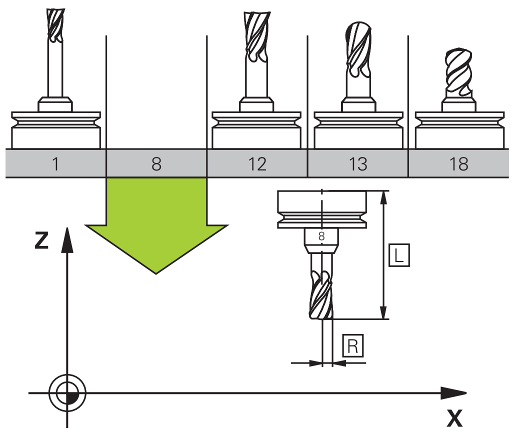
You can adjust the spindle speed during program run with the spindle speed potentiometer S.

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 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.

Assign unique tool names!

If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.

- Tool that is in the spindle
- Tool that is in the magazine

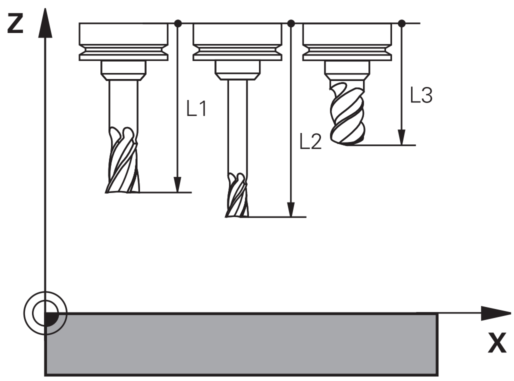
> Refer to your machine manual.
>
> If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

- Tool that is defined in the tool table but is currently not in the magazine

If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.

Tool length L

Always enter the tool length **L** as an absolute value based on the tool reference point.



> The absolute length of the touch probe is always referenced to the tool reference point. The machine manufacturer usually defines the spindle nose as the tool reference point.

Measuring the tool length

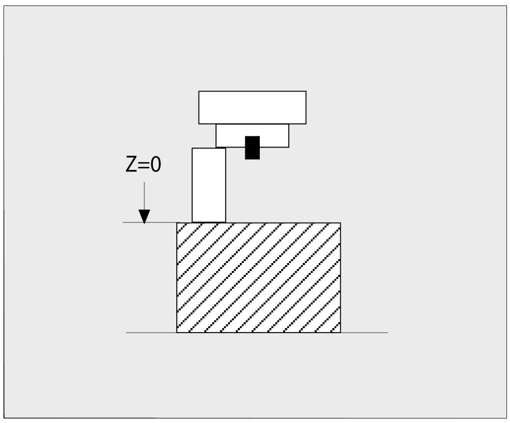
You can measure your tools in the machine (e.g., with a tool touch probe) or externally with a tool presetter. If such measurements are not possible, you can determine the tool length.

You have the following options for determining the tool length:

- With a gauge block
- With a calibration pin (inspection tool)

> Before you determine tool length, you have to set the preset in the spindle axis.

Determining the tool length with a gauge block



> You can only set the preset with a gauge block if the tool reference point is at the spindle nose.
>
> Place the preset on the surface you want to touch off with the tool. This surface might have to be created first.

To set the datum with a gauge block:

- Place the gauge block on the machine table
- Position the spindle nose next to the gauge block
- Gradually move in **Z+** direction until you can just slide the gauge block under the spindle nose
- Set the preset in **Z**

To determine the tool length:

- Insert the tool
- Touch off the surface
- > The control displays the absolute tool length as the actual position in the position display.

Determining the tool length with a calibration pin and a tool setter

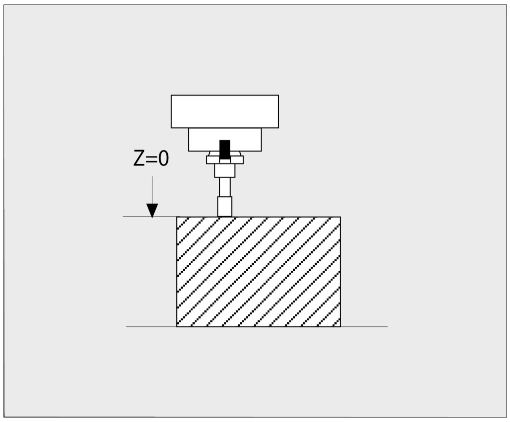
To set the preset with a calibration pin and a tool setter:

- Clamp the tool setter onto the machine table.
- Bring the flexible inner ring of the tool setter to the same height as the fixed outer ring.
- Set the gauge to 0
- Move the calibration pin onto the flexible inner ring.
- Set the preset in **Z**

To determine the tool length:

- Insert the tool
- Move the tool onto the flexible inner ring until the gauge displays 0.

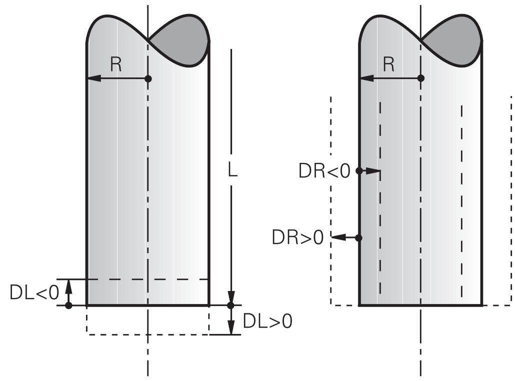
> The control displays the absolute tool length as the actual position in the position display.



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 represents a tool oversize (**DL**, **DR**>0). For a machining operation with an oversize, enter the value for the oversize in the NC program with **TOOL CALL** or with the help of a compensation table.

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 NC program do not change the depicted size of the **tool** in the simulation. However, the programmed delta values move the **tool** in the simulation by the amount of the defined value.

Tool-specific Q parameters used as delta values

The control calculates all tool-specific Q parameters while a tool call is being executed. The respective Q parameters cannot be used as delta values until the tool call has been completed.

Tool-specific Q parameters that can be used:

Q parameters	Function
Q108	ACTIVE TOOL RADIUS
Q114	ACTIVE TOOL LENGTH

To be able to use tool-specific Q parameters as delta values, you need to program a second tool call.

Example of ball-nose cutter:

You can use **Q108** (active tool radius) to correct the length of a ball-nose cutter to its center (**DL - Q108**).

```
1 TOOL CALL "BALL_MILL_D4" Z S10000
2 TOOL CALL  DL-Q108
```

Entering tool data into the NC program

Refer to your machine manual.
The machine manufacturer 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 call**: 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 puts the tool name in quotation marks. You must first assign a tool name to a string parameter. The names refer to an entry in the active tool table TOOL.T.

SELECT

- 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 tool radius 2

> The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
>
> Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

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, tool number, 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 of 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 manufacturer.

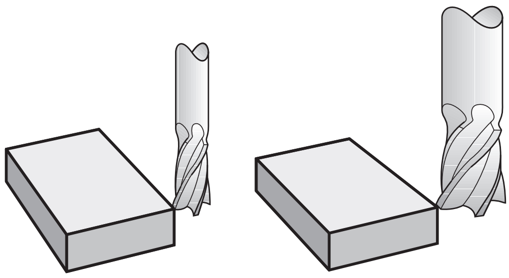
The tool condition at the end of planned tool life depends on e.g. the tool type, machining method and workpiece material. In the **OVRTIME** column of the tool table, enter the time in minutes for which the tool is permitted to be used beyond the tool life.

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 automatically becomes active as soon as a tool is called. It is canceled as soon as a tool is called with the length L=0 (e.g., **TOOL CALL 0**).

NOTICE

Danger of collision!

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

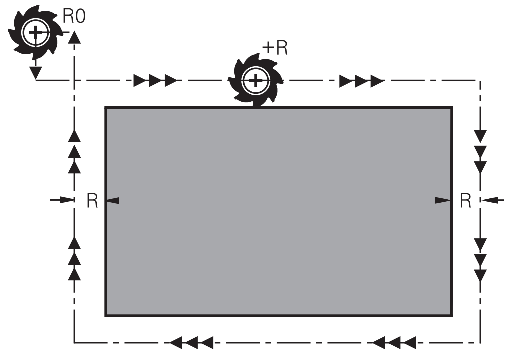
- Always define the actual tool length of a tool (not just the difference)
- Use **TOOL CALL 0** only to empty the spindle

With length compensation, delta values from both the NC program and the tool table are considered.

Compensation value = **L** + $\mathbf{DL}_{TAB}$ + $\mathbf{DL}_{Prog}$ with

L:	Tool length **L** from **TOOL DEF** block or tool table
$\mathbf{DL}_{TAB}$:	Oversize for length **DL** in the tool table
$\mathbf{DL}_{Prog}$:	Oversize **DL** for length from **TOOL CALL** block or from the compensation table The most recently programmed value becomes active. **Further information:** "Compensation table", Page 328

Tool radius compensation



An NC block can contain the following types of tool radius compensation:

- **R+** lengthens a paraxial movement by the amount of the tool radius
- **R-** shortens a paraxial movement by the amount of the tool radius
- **R0** positions the tool with the tool center

> The control shows an active tool compensation in the general status display.

The radius compensation takes effect as soon as a tool is called and is moved with one of the above-mentioned types of tool radius compensation within a paraxial movement in the working plane.

> Radius compensation is not active for positioning 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:

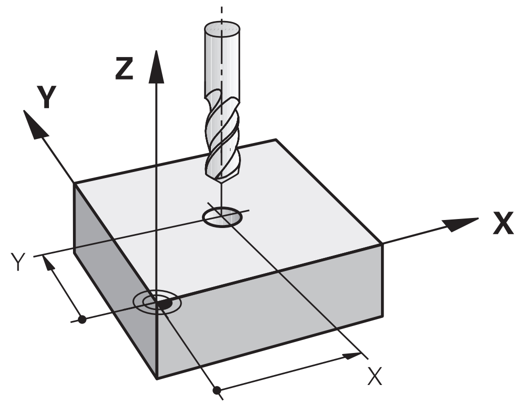
Compensation value = **R** + $\mathbf{DR}_{TAB}$ + $\mathbf{DR}_{Prog}$ with

R:	Tool radius **R** from **TOOL DEF** block or tool table
$\mathbf{DR}_{TAB}$:	Oversize **DR** for the radius in the tool table
$\mathbf{DR}_{Prog}$:	Oversize **DR** for the radius from the **TOOL CALL** block or from the compensation table **Further information:** "Compensation table", Page 328

Movements without radius compensation: R0

The tool center moves in the working plane to the programmed coordinate.

Applications: Drilling and boring, pre-positioning



Entering radius compensation within paraxial movements

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 lengthens the traverse path of the tool by the amount of the tool radius

R-
- The TNC shortens the traverse path of the tool by the amount of 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

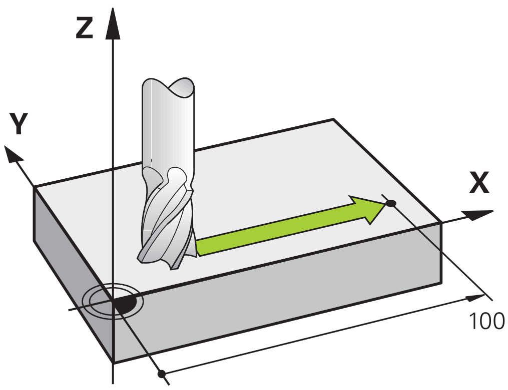
5

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", Page 125

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 177

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 3D touch probe during the program run.

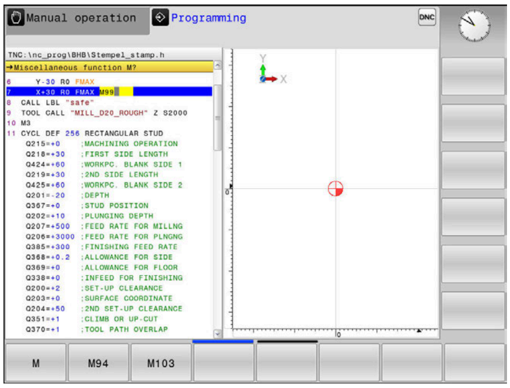
Further information: "Programming Q Parameters", Page 201

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
```

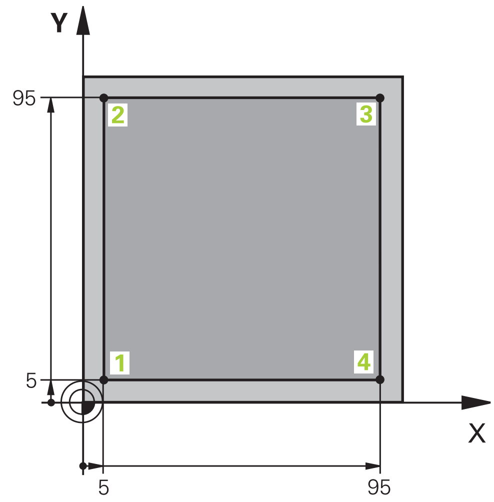
Capture actual position

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

- In **Manual operation** mode, move the tool to the position you want to capture
- 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 spindle speed
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:

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

To select special functions:

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

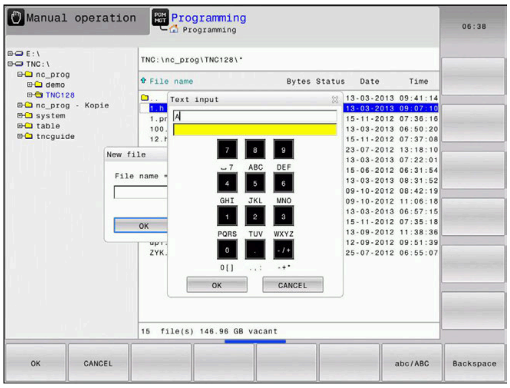
Further information: "Defining a cycle using the GOTO function", Page 350

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

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

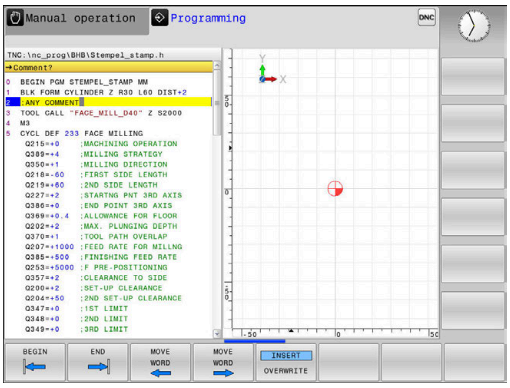
- Use the **OK** soft key to load the text into the open dialog field

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

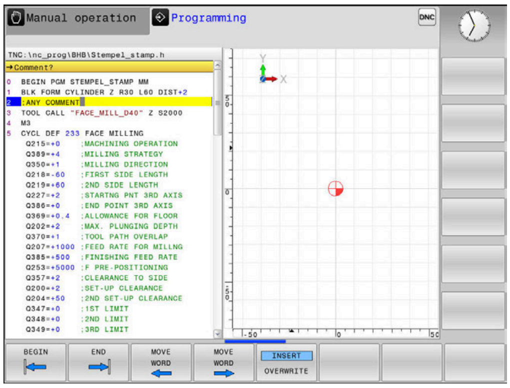
Scrollbar

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.



> 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 be a tilde sign (~).

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 **PROGRAM- MING AIDS** soft key

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

Entering comments during programming

> To use this function you will need a 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

> 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

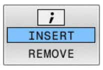
> To use this function you will need a alphabetic keyboard connected via USB.

- 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

To change an existing NC block into 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 into an NC block

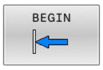
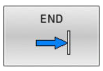
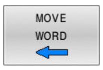
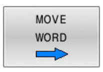
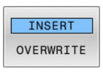
To change a commented-out NC block into an active NC block:

- 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 a 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 insert 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

To add syntax to an existing NC program:

- ▶ Press the **PGM MGT** key
- > The control opens the file manager.
- ▶ Press the **MORE FUNCTIONS** soft key
- ▶ Press the **SELECT EDITOR** soft key
- > The control opens a selection window.
- ▶ Select the **TEXT EDITOR** option
- ▶ Confirm your selection with **OK**
- ▶ Add the desired syntax

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

Free syntax input using the ? key in the NC editor

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

To add syntax to an existing, open NC program:

- ▶ Enter **?**
- > The control opens a new NC block.
- ▶ Add the desired syntax
- ▶ Confirm your entry with **END**

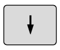
> After confirmation, the control checks the syntax. Errors will result in **ERROR** blocks.

6.6 Skipping NC blocks

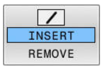
Insert a slash (/)

You can optionally hide NC blocks.

To hide NC blocks in the **Programming** mode:



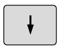
- Select the desired NC block



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

Delete the slash (/)

To show NC blocks again in the **Programming** mode:

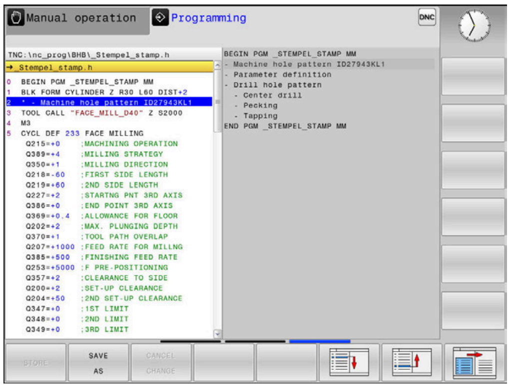


- Select the hidden NC block

- 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

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

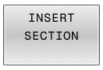
- Change the active window: Press the **CHANGE WINDOW** soft key

Inserting a structure block in the program window

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

- Press the **SPEC FCT** key

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



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

- Change the structuring depth (indenting) via soft key

> You can indent structure items only during editing.

Selecting blocks in the program structure window

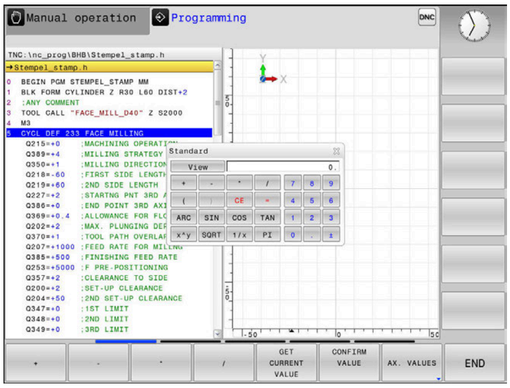
If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

6.8 Calculator

Operation

The control features a calculator with the most important mathematical functions.

- To show the calculator, press the **CALC** key
- Select the arithmetic functions: Select the command via soft key or enter it with an alphanumeric keyboard
- To close the calculator, press the **CALC** key



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

Calculator function	Command (soft key)
Truncate decimal places	INT
Truncate digits before the decimal point	FRAC
Modulo	MOD
Select view	View
Delete value	CE
Unit of measure	MM or INCH
Show angular value in radians (default: angular value in degrees)	RAD
Select numerical value notation	DEC (decimal) or HEX (hexadecimal)

Transferring the calculated value into the NC program

- With the arrow keys, select the word into which the calculated value is to be transferred
- Show the 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.

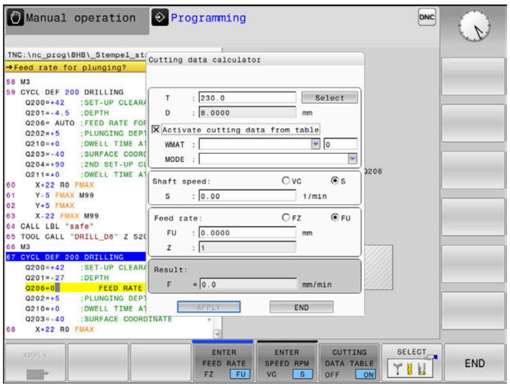
Functions in the pocket calculator

Soft key	Function
AX. VALUES	Transfer the nominal or reference value of the respective axis position into the calculator
GET CURRENT VALUE	Transfer the numerical value from the active input field into the calculator
CONFIRM VALUE	Transfer the numerical value from the calculator 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
- Define spindle speeds
- Define feed rates
- Press the **F** soft key in **Manual Operation** mode
- Press the **S** soft key in **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
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

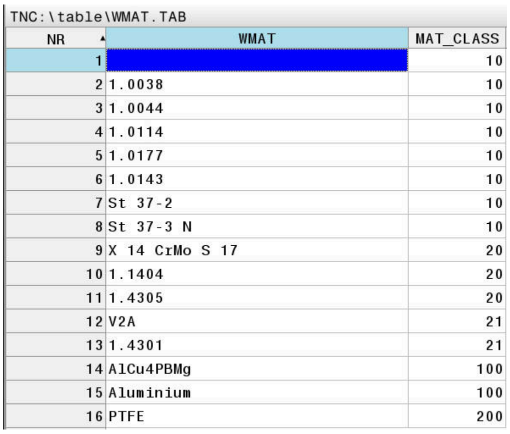
Application

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



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

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

This table contains the **WMAT** column for material and a **MAT_CLASS** column where you can categorize the materials by 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:

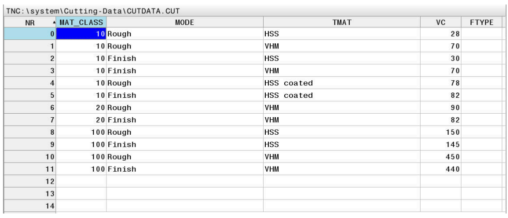
- Select the cutting data calculator
- Select **Activate cutting data from table** in the pop-up window
- Select **WMAT** from the selection menu

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



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					

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.

> Using the simplified cutting data table, you can determine speeds and feed rates using cutting data that are independent of the tool radius (e.g., **VC** and **FZ**).
>
> If you require specific cutting data depending on the tool radius for your calculations, use the diameter-dependent cutting data table.
>
> **Further information:** "Diameter-dependent cutting data table ", Page 149

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

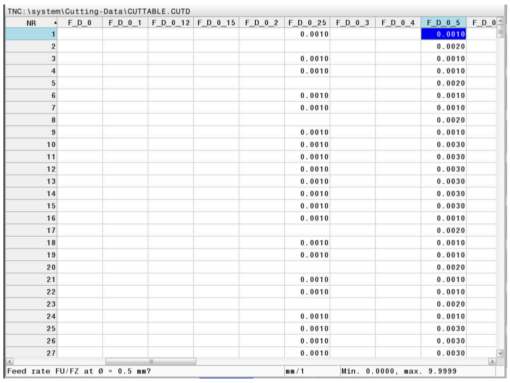
Diameter-dependent cutting data table

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

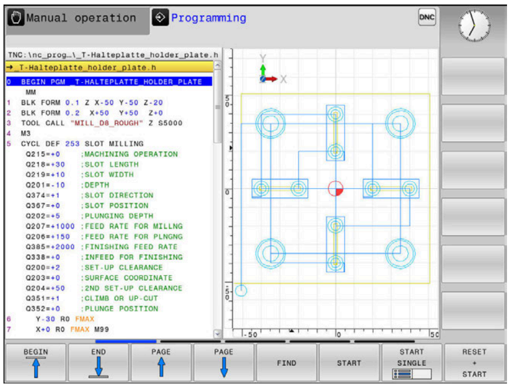
> You don't need to fill in all columns. If a tool diameter is between two defined columns, the control linearly interpolates the feed rate.

Note

In the corresponding folders, the control provides sample tables for automatic cutting data calculation. You can customize theses tables and specify your own data, i.e. materials and tools to be used.

6.10 Programming graphics

Activating and deactivating programming graphics



While you are writing an NC program, you can have the control generate a 2D 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 want the control to generate graphics during programming, then set the **AUTO DRAW** soft key to **OFF**.

> If **AUTO DRAW** is set to **ON**, then the control ignores the following program content when creating 2D 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 when you reopen an NC program or press the **RESET + START** soft key.

The control uses various colors in the programming graphics:

- **blue:** completely defined contour element
- **violet:** not yet completely defined 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 graphic: 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 programming 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 ■ Side view
SHOW TOOL PATHS OFF ON	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

SHOW BLOCK NO. OFF ON

- Show block numbers: Set the **SHOW BLOCK NO.** soft key to **ON**
- Hide block numbers: Set the **SHOW BLOCK NO.** soft key to **OFF**

Erasing the graphic

▷

- Shift the soft-key row

CLEAR GRAPHICS

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

Showing grid lines

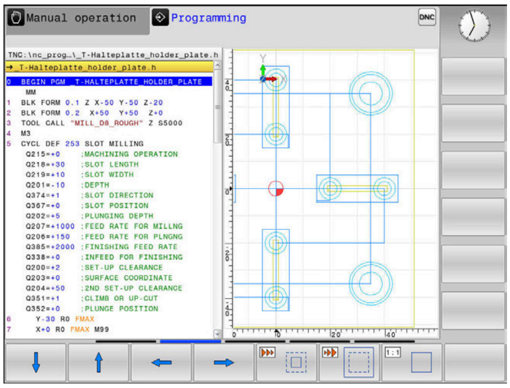
▷

- Shift the soft-key row

OFF ON

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

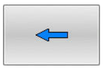
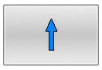
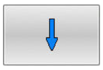
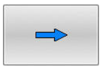
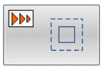
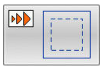
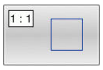
Magnification or reduction of details



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

The **RESET BLK FORM** soft key allows you to restore the original section.

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

- To shift the displayed model, hold down the center mouse button or the mouse wheel, and move the mouse. If you press the shift key at the same time, then you will be able to shift the model only 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 input
- Logical errors in the NC program
- Contour elements that are impossible to machine
- Incorrect use of touch probes
- Hardware updates

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

The control uses the following icons and text colors for different error classes:

Icon	Text color	Error class	Meaning
	Red	Error Prompt	The control displays a dialog with several options you can select from. **Further information:** "Detailed error messages", Page 154
	Red	Reset error	The control must be restarted. This message cannot be cleared.
	Red	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Yellow	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
	Blue	Information	You can continue without clearing the message. You can clear the information at any time.
	Green	Note:	You can continue without clearing the message. The control displays the note until you press the next valid key.

The table rows are ordered by priority. The control displays a message in the header until it is cleared or replaced by a higher-priority message (higher error class).

The control displays long and multi-line error messages in abbreviated form. The complete information on all pending errors is shown in the error window.

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.

Opening the error window

When you open the error window, the complete information on all pending errors will be shown.

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

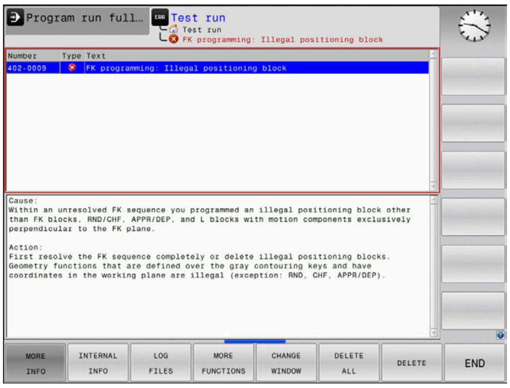
Detailed error messages

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

- Open the error window
- Position the cursor on the corresponding error message

- Press the **MORE INFO** soft key
- > The control opens a window with information on the error cause and corrective action.

- Exit the info: Press the **MORE INFO** soft key again



High-priority error messages

When an error message occurs at switch-on of the control due to hardware changes or updates, the control will automatically open the error window. The control displays an error of the question type.

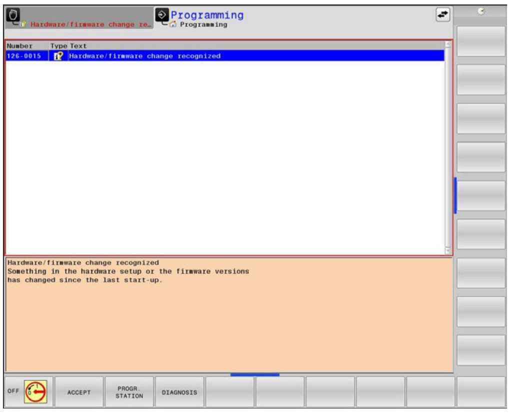
You can correct this error only by pressing the corresponding soft key to acknowledge the question. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.

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

If a rare **processor check error** should occur, the control will automatically open the error window. You cannot correct such an error.

Proceed as follows:

- Shut down the control
- Restart



INTERNAL INFO soft key

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

- Open the error window
- Position the cursor on the corresponding error message

- Press the **INTERNAL INFO** soft key
- > The control opens a window with internal information about the error.

- Exit the detailed information: Press the **INTERNAL INFO** soft key again

GROUPING soft key

If you activate the **GROUPING** soft key, the control displays all warnings and error messages with the same error number in the same line of the error window. This makes the list of messages shorter and easier to read.

To group the error messages:

ERR

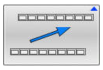
- ▶ Open the error window

MORE FUNCTIONS

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

GROUPING OFF ON

- ▶ Press the **GROUPING** soft key
- > The control groups identical warnings and error messages.
- > The number of occurrences of the individual messages is indicated in parentheses in the respective line.



- ▶ Press the **GO BACK** soft key

ACTIVATE AUTOMATIC SAVING soft key

The **ACTIVATE AUTOMATIC SAVING** soft key allows you to specify error numbers that cause the control to save a service file if an error with that number occurs.

ERR

- ▶ Open the error window

MORE FUNCTIONS

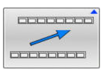
- ▶ Press the **MORE FUNCTIONS** soft key

ACTIVATE AUTOMATIC SAVING

- ▶ Press the **ACTIVATE AUTOMATIC SAVING** soft key
- > The control opens the **ACTIVATE AUTOMATIC SAVING** pop-up window.
- ▶ Define the entries
  - **Error number**: Enter the desired error number
  - **active:** Enable this option to automatically create the service file
  - **Comment:** Enter a comment on this error number, if required

STORE

- ▶ Press the **STORE** soft key
- > If an error with the specified error number occurs, a service file will be saved automatically.



- ▶ Press the **GO BACK** soft key

Deleting errors

The control can automatically clear pending warning or error messages when an NC program is selected or restarted. The machine manufacturer specifies in the optional machine parameter **CfgClearError** (no. 130200) whether these messages will automatically be cleared.

The factory default setting of the control defines that warning and error messages in the **Test Run** and **Programming** operating modes will be cleared automatically from the error window. Messages issued in the machine operating modes will not be cleared.

Clearing errors outside of the error window

CE

- Press the **CE** key
- > The control clears the errors or notes being displayed in the header.

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
- Position the cursor on the corresponding error message

DELETE

- Press the **DELETE** soft key

DELETE ALL

- As an alternative, clear all errors: 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. When the log is full, the control uses a second file. When 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

- Press the **LOG FILES** soft key

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

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

- 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. When the keystroke log is full, the control switches to a second keystroke log. When 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

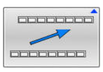
- Set the previous keystroke log if required: Press the **PREVIOUS FILE** soft key

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 situation 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).

> In order to facilitate sending service files via email, the control will only save active NC programs with a size of up to 10 MB in the service file. If the NC program is larger, it will not be added to the created service file.

When the same name is entered several times in the **SAVE SERVICE FILES** function, the control saves up to five files and deletes the file with the oldest timestamp, as needed. Make a backup of the service files you created (e.g., by moving them to a different folder).

Saving service files

ERR
- 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
- Press the **OK** soft key
- > The control saves the service file.

Closing the error window

To close the error window:

END
- Press the **END** soft key

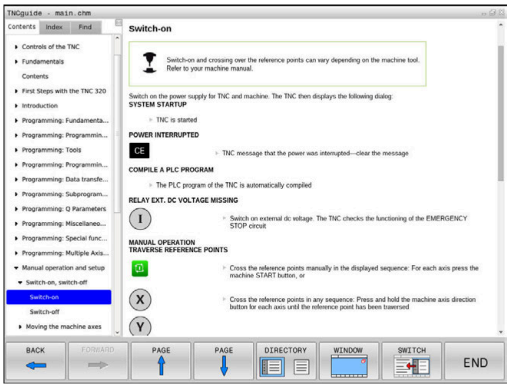
ERR
- Alternative: Press the **ERR** key
- > The control closes the error window.

6.12 TNCguide: context-sensitive help

Application

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

Further information: "Downloading current help files", Page 165



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

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

The following user documentation is available in **TNCguide**:

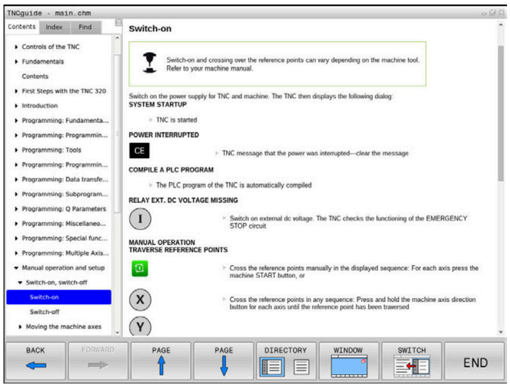
- User's Manual for Klartext Programming (**BHBKlartext.chm**)
- User's Manual for Setup, Testing and Running NC Programs (**BHBoperate.chm**)
- List of All Error Messages **(errors.chm)**

In addition, the **main.chm** "book" file is available, in which all existing .chm files are shown in one place.

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

Using TNCguide

Calling TNCguide



You have several options for starting **TNCguide**:

- Use the **HELP** key
- First click the help symbol in the lower right-hand corner of the screen, then click the appropriate soft key
- Open a help file (.chm file) via the file management. The control can open any .chm file, even if it is not saved in the control's internal memory

> On the Windows programming station, **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 **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 TNCguide

It's easiest to use the mouse to navigate in **TNCguide**. A table of contents appears on the left side of the screen. Clicking on the rightward pointing triangle opens subordinate sections, and clicking on the respective entry opens the corresponding page. You can use it in the same way as 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.

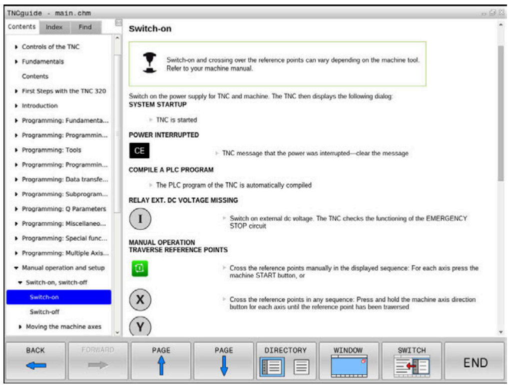
Soft key/ Keys	Function
↑ ↓	▪ If the table of contents at left is active: Select the entry above it or below it ▪ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
→	▪ If the table of contents at left is active: Expand the table of contents ▪ If the text window at right is active: No function
←	▪ If the table of contents at left is active: Collapse 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 right side of the window ▪ If the text window at right is active: Jump back to the left side of the window
	▪ 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 ↑	Go back one page
PAGE ↓	Go forward one page

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

Subject index

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

The left side is active.



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

Alternative:

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

> **i** You can enter the search word only with 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 website:
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, such as TNC 128 (77184x-18)

> HEIDENHAIN has simplified the version schema, starting with NC software version 16:
> - The publication period determines the version number.
> - All control models of a publication period have the same version number.
> - The version number of the programming stations corresponds to the version number of the NC software.

- Select the desired language version from the **TNCguide online help (CHM files)** 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**

Language	TNC directory
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. With some miscellaneous functions, the dialog is extended so that you can enter the required parameters for this function.

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

Effectiveness of miscellaneous functions

Some miscellaneous functions take effect at the start of the NC block and others at the end, regardless of the sequence in which they were programmed.

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

Some miscellaneous functions are effective block-by-block, i.e. only in the NC block in which the miscellaneous function has been programmed. When a miscellaneous function takes effect modally, you have to cancel this miscellaneous function again in a subsequent NC block (e.g., by using **M9** to switch off coolant that was switched on with **M8**). If miscellaneous functions are still active at the end of the program, the control will rescind the miscellaneous functions.

> If multiple M 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.

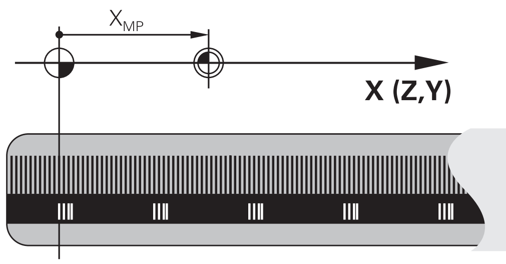
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine manufacturer)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 0 Clear status display Functional scope depends on machine parameter **resetAt** (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle 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 (such as 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 referenced to the machine datum, enter M91 into these NC blocks.

> If you program incremental coordinates in an NC block with the miscellaneous function **M91**, then these coordinates are relative to the last position programmed with **M91**.
> If the active NC program does not contain a position programmed with **M91**, the coordinates reference the current tool position.

The coordinate values on the control's screen are referenced to 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 manufacturer can also define an additional machine-based position as a reference point (machine preset).
> For each axis, the machine manufacturer defines the distance between the machine preset 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.

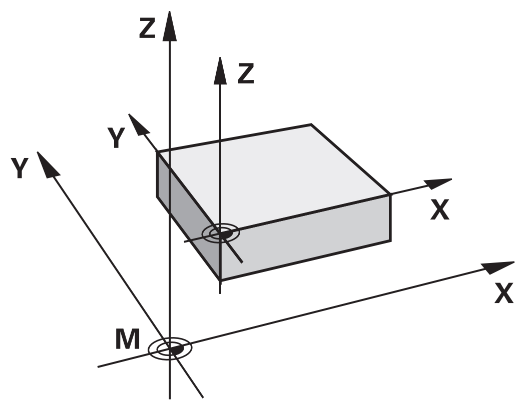
M91 and M92 take effect at the start of block.

Workpiece preset

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

If presetting is inhibited for all axes, the control does not display the **SET PRESET** soft key in the **Manual operation** operating 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

M94 only affects rollover axes whose actual position display permits values above 360°.

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

> Refer to your machine manual.
> In the machine parameter **isModulo** (no. 300102) the machine manufacturer defines whether the modulo counting method is used for a rollover axis.

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.

21 L M94	; Reduce the display values of all rotary axes
21 L M94 C	; Reduce the display value of the C axis
21 L C+180 FMAX M94	; Reduce the display values of all active rotary axes and then move in the C axis to the programmed value

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 **FU** or **FZ**.
>
> The workpiece spindle is not permitted to be controlled when **M136** is active.
>
> It is not possible to combine **M136** with an oriented spindle stop. The control cannot calculate the feed rate because the spindle does not rotate during an oriented spindle stop.

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 the optional machine parameter **moveBack** (no. 200903), the machine manufacturer defines how far before a limit switch or a collision object a retraction movement **MB MAX** should end.

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 it is 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** the control retracts the tool only in the positive direction in the tool axis.
>
> The control gleans the necessary information about the tool axis for **M140** from the tool call.

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

Subprograms and program section repeats start with **LBL** in the NC program (an abbreviation for LABEL).

A LABEL contains a number between 1 and 65535 or a name to be defined by you. LABEL names can have up to 32 characters.

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

You may assign each LABEL number, or each LABEL name, only once in the NC program using the **LABEL SET**. The quantity of label names that may be entered is limited only by the amount of internal memory.

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

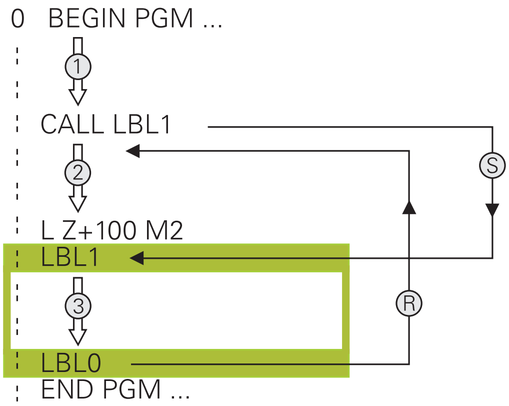
Label 0 (**LBL 0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

> Before creating your NC program, compare the subprogram and program section repeat programming techniques using if-then decisions.
>
> You can thereby avoid possible misunderstandings and programming errors.
>
> **Further information:** "If-then decisions with Q parameters", Page 215

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

LBL SET

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

Calling a subprogram

LBL CALL

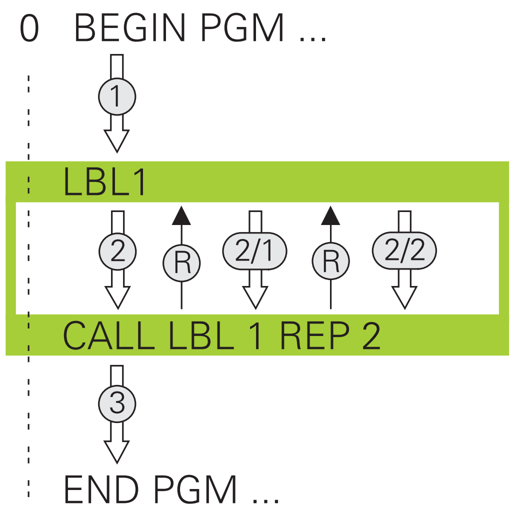
- Call a subprogram: Press the **LBL CALL** key
- Enter the subprogram number of the subprogram you wish to call. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- If you want to enter the number of a string parameter as target address, press the QS soft key
- > 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 Calling an external NC program

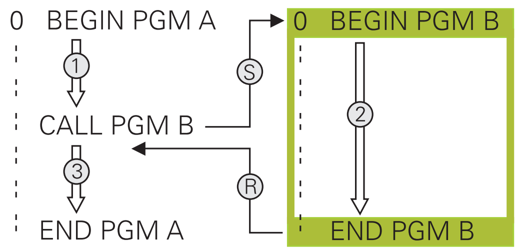
Overview of the soft keys

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

Soft key	Function	Description
CALL PROGRAM	Call an NC program with **CALL PGM**	Page 186
SELECT DATUM TABLE	Select a datum table with **SEL TABLE**	Page 327
SELECT POINT TABLE	Select a point table with **SEL PATTERN**	Page 190
SELECT PROGRAM	Select an NC program with **SEL PGM**	Page 187
CALL SELECTED PROGRAM	Call the last selected file with **CALL SELECTED PGM**	Page 187
SELECT CYCLE	Select any NC program with **SEL CYCLE** as a machining cycle	Page 353

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 an NC program.
- The called NC program must not use **CALL PGM** to call the calling NC program (an endless loop would ensue).
- The called NC program must not contain the miscellaneous function **M2** or **M30**. If you have defined subprograms with labels in the called NC program, then you can replace M2 or M30 with the jump function **FN 9: If +0 EQU +0 GOTO LBL 99**.
- If you want to call an 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 **Select the cycle** function (**SEL CYCLE**).
- As a rule, Q parameters are globally effective when used with a program call, such as **CALL PGM**. So please note that changes made to Q parameters in the called NC program also influence the calling NC program. If applicable, use QL parameters that take effect only in the active NC program.

> While the control is executing the calling NC program, editing of all called NC programs is disabled.

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 for completeness before execution. 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\PGM2.H**
- Starting from the folder of the calling NC program, one folder level up and into another folder **..\THERE\PGM3.H**

Use the **SYNTAX** soft key to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path.

Further information: "File names", Page 97

If the complete path is enclosed in quotation marks, you can use both **** and **/** to separate the folders and files.

Calling an external NC program

Calling with CALL PGM

Use the NC function **CALL PGM** to call an external NC program. The control runs the external NC program from the position where it was called in the NC program.

Proceed as follows:

- Press the **PGM CALL** key

- 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:

- 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

> If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

Call with SEL PGM and CALL SELECTED PGM

The function **SEL PGM** allows you to select an external NC program that you can separately call at a different position in the NC program. The control runs the external NC program from the position at which you called it in the NC program using **CALL SELECTED PGM**.

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

To select the NC program:

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

> If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

To call the selected NC program:

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 NO110** and **NO111**) to check all paths at the beginning of the program.
> **Further information:** "FN 18: SYSREAD – Reading system data", Page 243

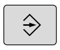
8.5 Point tables

Application

With a point table you can execute one or more cycles in sequence on an irregular point pattern.

Creating a point table

To create a point table:

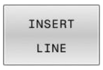


- Select the **PROGRAMMING** operating mode

- Press the **PGM MGT** key
- \> The control opens the file manager.
- Select the desired folder in your folder structure
- Enter the name and file type (*.**pnt**)

- Confirm with the **ENT** key

- Press the **MM** or **INCH** soft key.
- \> The control opens the table editor and shows an empty point table.



- Press the **INSERT LINE** soft key
- \> The control inserts a new row in the point table.
- Enter the coordinates of the desired machining position
- Repeat the process until all desired coordinates have been entered

> If you intend to use the point table in SQL queries later, the table name must begin with a letter.

Configuring the point table display

To configure the display of a point table:

- Open the desired point table

  Further information: "Creating a point table", Page 188
- Press the **SORT/ HIDE COLUMNS** soft key
- > The control opens the **Column sequence** window.
- Configure how the table will be displayed
- Press the **OK** soft key
- > The control will display the table as defined in the selected configuration.

> If you enter the code number 555343, the control will display the **EDIT FORMAT** soft key. With this soft key, you can change the table properties.

Hiding single points for 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.

To hide points:

- Select the desired point from the table
- Select the **FADE** column
- Activate hiding with the **ENT** key
- Deactivate hiding with the **NO ENT** key

Selecting a point table in the NC program

To select a point table in your NC program:

- In the **Programming** operating mode, select the NC program for which you want to activate the point table.

PGM CALL

- Press the **PGM CALL** key

SELECT POINT TABLE

- Press the **SELECT POINT TABLE** soft key

SELECT FILE

- Press the **SELECT FILE** soft key
- Select the point table from the folder structure
- Press 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.

> If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

Example

```
7 SEL PATTERN  "TNC:\nc_prog\Positions.PNT"
```

Using point tables

To call a cycle at the points defined in the point table, program the cycle call with **CYCL CALL PAT**.

With **CYCL CALL PAT**, the control will process the point table that you defined last.

To use a point table:

CYCL CALL

- Press the **CYCL CALL** key

CYCLE CALL PAT

- Press the **CYCL CALL PAT** soft key
- Enter the feed rate, e.g. **F MAX**

> The control will use this feed rate to traverse between the points of the point table. If you do not define a feed rate, the control will use the feed rate that was defined last.

- Enter a miscellaneous function if required
- Press the **END** key

Notes

- In the **GLOBAL DEF 125** function you can use the setting **Q435**=**1** to force the control to always move to the 2nd set-up clearance from the cycle during the positioning between the points.
- If you want to move at reduced feed rate when pre-positioning in the tool axis, program the **M103** miscellaneous function.
- With **CYCL CALL PAT** the control runs the point table that you last defined, even if you defined the point table with an NC program that was nested with **CALL PGM**.

Definition

File type	Definition
***.pnt**	Points table

8.6 Nesting

Types of nesting

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

> Subprograms and program-section repeats can call external NC programs as well.

Nesting depth

The nesting depth defines, among other things, how often program sections or subprograms may contain further subprograms or program section repeats.

- Maximum nesting depth for subprograms: 19
- Maximum nesting depth for external NC programs: 19, for which a **CYCL CALL** has the effect of calling an external program
- 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 UP1
...	
35 Z+100 R0 FMAX M2	Last program block of the main program with M2
36 LBL "UP1"	Beginning of subprogram UP1
...	
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. Program end and return jump to NC block 0

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. Program end and return jump to NC block 0

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

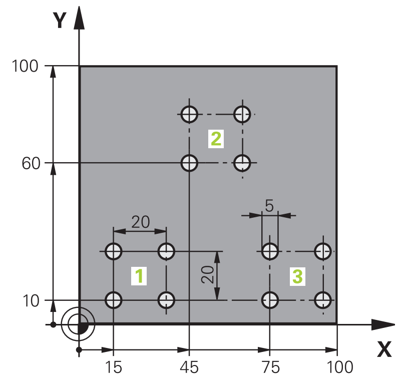
1 Main program UPGREP is executed up to NC block 11
2 Subprogram 2 is called and executed.
3 The program section between NC block 12 and NC block 10 is repeated twice. This means that subprogram 2 is repeated twice
4 Main program UPGREP is executed from NC block 13 up to NC block 19. Program end and return jump to NC block 0

8.7 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



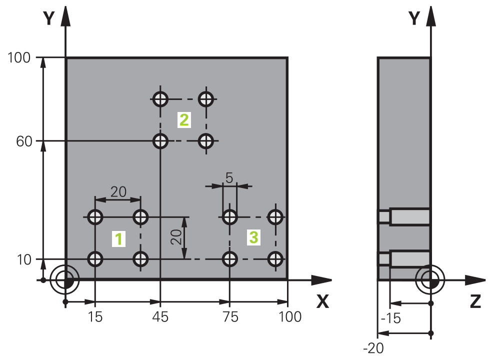
0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3000	Tool call
4 Z+250 R0 FMAX M3	
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q210=+0 ;DWELL 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 multiple 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



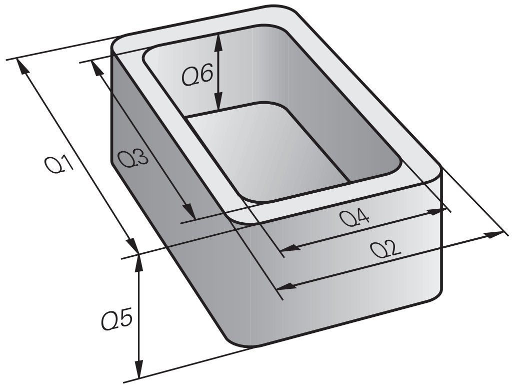
0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Centering drill tool call
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: Centering
Q200=2 ;SET-UP CLEARANCE	
Q201=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=3 ;PLUNGING 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	Hole 1 with active machining 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.

Q parameters can be used in the following ways:

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

The control offers more ways to use Q parameters:

- Program contours that are defined through mathematical functions
- Making the execution of machining steps dependent on logical conditions

Q parameter types

Q parameters for numerical values

Variables always consist of letters and numbers. The letters determine the type of variable and the numbers its range.

For more information, see the table below:

Variable type	Variable range	Meaning
Q parameters:		Q parameters affect all NC programs in the control's memory.
	0 to 99	User-defined Q parameters, if there are no overlaps with the HEIDENHAIN SL cycles Q parameters between 0 and 99 have a local effect within macros and cycles. This means that the control will not return changes to the NC program. For this reason, use the Q parameter range 1200 to 1399 for machine manufacturer cycles!
	100 to 199	Q parameters for special functions on the control that can be read by user-defined NC programs or by cycles
	200 to 1199	Q parameters for functions defined by HEIDENHAIN (e.g., cycles)
	1200 to 1399	Q parameters for functions defined by the machine manufacturer (e.g., cycles)
	1400 to 1999	User-defined Q parameters
QL parameters:		QL parameters are active locally within an NC program.
	0 to 499	User-defined QL parameters
QR parameters:		QR parameter affect all NC programs in the control's memory; they are retained even after a restart of the control.
	0 to 99	User-defined QR parameters
	100 to 199	QR parameters for functions defined by HEIDENHAIN (e.g., cycles)
	200 to 499	QR parameters for functions defined by the machine manufacturer (e.g., cycles)

QR parameters will be included in backups.

If the machine manufacturer did not define a specific path, the control saves the QR parameters in the following path: **SYS:\runtime\sys.cfg**. The **SYS:** partition will only be backed up in full backups.

Machine manufacturers can use the following optional machine parameters to specify the paths:

- **pathNcQR** (no. 131201)
- **pathSimQR** (no. 131202)

If the machine manufacturer used the optional machine parameters to specify a path on the **TNC:** partition, you can perform a backup with the **NC/PLC Backup** functions without entering a code number.

Q parameters for texts

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

The following characters can be used within QS parameters:

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 ; ! # $ % & ' ( ) + , - . / : < = > ? @ [ ] ^ _ ` *

Variable type	Variable range	Meaning
QS parameters:		QS parameters affect all NC programs in the control’s memory.
	0 to 99	User-defined QS parameters, if there are no overlaps with the HEIDENHAIN cycles QS parameters between 0 and 99 have a local effect within macros and cycles. This means that the control will not return changes to the NC program. For this reason, use the QS parameter range 1200 to 1399 for machine manufacturer cycles!
	100 to 199	QS parameters for special functions on the control that can be read by user-defined NC programs or by cycles
	200 to 1199	QS parameters for functions defined by HEIDENHAIN (e.g., cycles)
	1200 to 1399	QS parameters for functions defined by the machine manufacturer (e.g., cycles)
	1400 to 1999	User-defined QS parameters

Programming notes

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 manufacturer, and suppliers.
- Check the machining sequence using a graphic simulation

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

Variables 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. The control can calculate numerical values up to 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 260

The control saves numerical values internally in a binary number format (standard IEEE 754). Due to the standardized format used, some decimal numbers cannot be represented with a binary value that is 100% exact (rounding error). If you use calculated variable values for jump commands or positioning moves, you must keep this in mind.

Using the **SET UNDEFINED** syntax element, you can assign the **undefined** status to your variables. For example, if you program a position using an undefined Q parameter, the control will ignore this movement. If you use an undefined Q parameter in the calculation steps of your NC program, the control will display an error message and stop the program run.

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)	208
TRIGO-NOMETRY	Trigonometric functions	212
CIRCLE CALCU-LATION	Function for calculating circles	214
JUMP	If/then conditions, jumps	215
DIVERSE FUNCTION	Other functions	226
FORMULA	Entering formulas directly	218

> 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** allows you to assign numerical values to Q parameters. You then use a Q parameter in place of the numerical value in the NC program.

Example

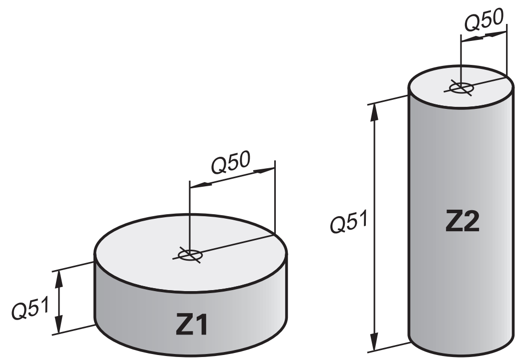
15 FN 0: Q10=25	Assign
...	Q10 is assigned the value 25
25 X +Q10	Means X +25

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 = Q50
Cylinder height:	H = Q51
Cylinder Z1:	Q50 = +30
	Q51 = +10
Cylinder Z2:	Q50 = +10
	Q51 = +50



9.3 Describing contours with mathematical functions

Application

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

- Select the Q parameter function: Press the **Q** key in the numeric keypad
- > The Q parameter functions are displayed in the soft key row.

- Press the **BASIC ARITHM.** soft key
- > The control displays the soft keys for basic mathematical functions

Overview

Soft key	Function
FN0 X = Y	**FN 0**: Assignment Example: **FN 0: Q5 = +60** Q5 = 60 Assign a value or the **Undefined** status
FN1 X + Y	**FN 1**: Addition Example: **FN 1: Q1 = -Q2 + -5** Q1 = –Q2+(–5) Calculate and assign the sum of two values
FN2 X - Y	**FN 2**: Subtraction Example: **FN 2: Q1 = +10 - +5** Q1 = +10–(+5) Calculate and assign the difference of two values.
FN3 X * Y	**FN 3**: Multiplication Example: **FN 3: Q2 = +3 * +3** Q2 = 3*3 Calculate and assign the product of two values.
FN4 X / Y	**FN 4**: Division Example: **FN 4: Q4 = +8 DIV +Q2** Q4 = 8/Q2 Calculate and assign the quotient of two values Restriction: You cannot divide by 0
FN5 SQRT	**FN 5**: Square root Example: **FN 5: Q20 = SQRT 4** Q20 = √4 Calculate and assign the square root of a number Restriction: You cannot calculate a square root from a negative value

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

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

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

Programming fundamental operations

Example: Assignment

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

Q

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

BASIC ARITHM.

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

FN0 X = Y

- To select the **ASSIGN** Q parameter function: Press the **FN 0 X = Y** soft key
- > The control asks you for the number of the result parameter.
- Enter **5** (number of Q parameter)

ENT

- Confirm with the **ENT** key
- > The control asks you for the value or parameter.
- Enter **10** (value)

ENT

- Confirm with the **ENT** key
- > As soon as the control reads the NC block, the value **10** is assigned to the parameter **Q5**.

Example: Multiplication

Q

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

BASIC ARITHM.

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

FN3 X * Y

- To select the **MULTIPLICATION** Q parameter function, press the **FN 3 X * Y** soft key
- > The control asks you for the number of the result parameter.
- Enter **12** (number of Q parameter)

ENT

- Confirm with the **ENT** key
- > The control asks you for the first value or parameter.
- Enter **Q5** (parameter)

ENT

- Confirm with the **ENT** key
- > The control asks you for the second value or parameter.
- Enter **7** for the second value

ENT

- 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.

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

FN0 X = Y

- To select the ASSIGN Q parameter function: Press the **FN 0 X = Y** soft key
- > The control asks you for the number of the result parameter.
- Enter **5** (number of Q parameter)

ENT

- Confirm with the **ENT** key
- > The control asks you for the value or parameter.

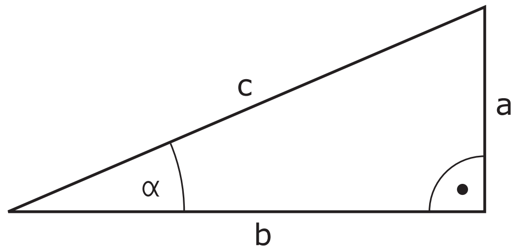
SET UNDEFINED

- Press **SET UNDEFINED**

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

9.4 Trigonometric functions

Definitions



Sine: $\sin \alpha$ = opposite side/hypotenuse
$\sin \alpha = a/c$

Cosine: $\cos \alpha$ = adjacent side/hypotenuse
$\cos \alpha = b/c$

Tangent: $\tan \alpha$ = opposite side/adjacent side
$\tan \alpha = a/b$ or $\tan \alpha = \sin \alpha/\cos \alpha$

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:

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

Example:

a = 25 mm

b = 50 mm

$\alpha = \arctan(a/b) = \arctan 0.5 = 26.57°$

Furthermore:

$a^2+b^2 = c^2$ (where $a^2 = a*a$)

$c = \sqrt{(a^2+b^2)}$

Programming trigonometric functions

You can also calculate trigonometric functions with Q parameters.

- ▶ Select the Q parameter function: Press the **Q** key in the numeric keypad
- > The Q parameter functions are displayed in the soft key row.

- ▶ Press the **TRIGO- NOMETRY** soft key
- > The control displays the soft keys for trigonometric functions.

Overview

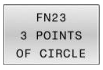
Soft key	Function
FN6 SIN(X)	**FN 6**: Sine Example: **FN 6: Q20 = SIN -Q5** Q20 = sin(–Q5) Calculate and assign the sine of an angle in degrees
FN7 COS(X)	**FN 7**: Cosine Example: **FN 7: Q21 = COS -Q5** Q21 = cos(–Q5) Calculate and assign the cosine of an angle in degrees
FN8 X LEN Y	**FN 8**: Root of the sum of squares Example: **FN 8: Q10 = +5 LEN +4** Q10 = $\sqrt{(5^2+4^2)}$ Calculate and assign the length based on two values (e.g., to calculate the third side of a triangle).
FN13 X ANG Y	**FN 13**: angle Example: **FN 13: Q20 = +25 ANG -Q1** Q20 = arctan(25/–Q1) Calculate and assign the angle from the opposite side and the adjacent side using arctan or from 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.

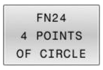
Application: These functions can be used, for example, if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Soft key	Function
	**FN 23**: Circle data from three points on the circle Example: **FN 23: Q20 = CDATA Q30** The control saves the determined values in the Q parameters **Q20** to **Q22**.

The control checks the values in the Q parameters **Q30** to **Q35** and determines the circle data.

The control saves the results in the following Q parameters:

- Circle center on the main axis in the Q parameter **Q20**
  For the tool axis **Z**, the main axis is **X**
- Circle center on the secondary axis in the Q parameter **Q21**
  For the tool axis **Z**, the secondary axis is **Y**
- Circle radius in the Q parameter **Q22**

Soft key	Function
	**FN 24**: Circle data from four points on the circle Example: **FN 24: Q20 = CDATA Q30** The control saves the determined values in the Q parameters **Q20** to **Q22**.

The control checks the values in the Q parameters **Q30** to **Q37** and determines the circle data.

The control saves the results in the following Q parameters:

- Circle center on the main axis in the Q parameter **Q20**
  For the tool axis **Z**, the main axis is **X**
- Circle center on the secondary axis in the Q parameter **Q21**
  For the tool axis **Z**, the secondary axis is **Y**
- Circle radius in the Q parameter **Q22**

> **FN 23** and **FN 24** not only assign a value to the results variable to the left of the equal sign, but also to the subsequent variables.

9.6 If-then decisions with Q parameters

Application

In if-then decisions, the control compares a variable or fixed value with another variable or fixed value. If the condition is fulfilled, the control jumps to the label programmed for the condition.

> Before creating your NC program, compare the if-then decisions with the subprogram and program section repeat programming techniques.
>
> You can thereby avoid possible misunderstandings and programming errors.
>
> **Further information:** "Labeling subprograms and program section repeats", Page 178

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

If you want to call an external NC program, then program a program call with **CALL PGM** after the label.

Abbreviations used

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

Jump conditions

Unconditional jump

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

FN 9: IF+10 EQU+10 GOTO LBL1

You can use such jumps, for example, in a called NC program in which you work with subprograms. In an NC program without **M30** or **M2**, you can prevent the control from executing subprograms without a call with **LBL CALL**. As the jump address, program a label that is located directly before the program end.

Conditioning jumps with counters

The jump function allows you to repeat a machining operation any number of times. A Q parameter serves as a counter that increments by 1 at every program section repeat.

The jump function allows you to compare the counter with the number of desired machining operations.

> These jumps differ from the subprogram and program section repeat programming techniques.
>
> On the one hand, for example, jumps require no completed program section ending with LBL 0. On the other hand, jumps do not take these return jump labels into consideration!

Example

0 BEGIN PGM COUNTER MM	
1 ;	
2 Q1 = 0	Loaded value: Initialize counter
3 Q2 = 3	Loaded value: Number of jumps
4 ;	
5 LBL 99	Label
6 Q1 = Q1 + 1	Initialize counter: New Q1 value = Old Q1 value + 1
7 FN 12: IF +Q1 LT +Q2 GOTO LBL 99	Run program jumps 1 and 2
8 FN 9: IF +Q1 EQU +Q2 GOTO LBL 99	Run program jump 3
9 ;	
10 END PGM COUNTER MM	

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**

The if-then decisions appear when the **JUMP** soft key is pressed. The control displays the following soft keys:

Soft key	Function
FN9 IF X EQ Y GOTO EQU	**FN 9**: jump if equal Example: **FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"** If both values are equal, the control jumps to the defined label.
FN9 IF X EQ Y GOTO IS UNDEFINED	**FN 9**: jump if undefined Example: **FN 9: IF +Q1 IS UNDEFINED GOTO LBL "UPCAN25"** If the variable is undefined, the control jumps to the defined label.
FN9 IF X EQ Y GOTO IS DEFINED	**FN 9**: jump if defined Example: **FN 9: IF +Q1 IS DEFINED GOTO LBL "UPCAN25"** If the variable is defined, the control jumps to the defined label.
FN10 IF X NE Y GOTO	**FN 10**: jump if not equal Example: **FN 10: IF +10 NE -Q5 GOTO LBL 10** If both values are not equal, the control jumps to the defined label.
FN11 IF X GT Y GOTO	**FN 11**: jump if greater than Example: **FN 11: IF+Q1 GT+10 GOTO LBL QS5** If the first value is greater than the second value, the control jumps to the defined label.
FN12 IF X LT Y GOTO	**FN 12**: jump if less than Example: **FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME"** If the first value is less than the second value, the control jumps to the defined label.

9.7 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 function

FORMULA

- Press the **FORMULA** soft key
- Select **Q**, **QL**, or **QR**
- > The control displays the available mathematical operations in the soft-key row.

Calculation rules

Evaluation order for different operators

If a formula includes arithmetic operations involving a combination of different operators, the control evaluates the operations in a certain order. A familiar example of this is the rule that multiplication/division takes precedence over addition/subtraction (higher-level operations are performed first).

The control evaluates the arithmetic operations in the following order:

Order	Arithmetic operation	Operator	Arithmetic operator
1	Perform operations in parentheses first	Parentheses	**( )**
2	Note the algebraic sign	Algebraic sign	**-**
3	Calculate functions	Function	**SIN**, **COS**, **LN**, etc.
4	Exponentiation	Power	**^**
5	Multiplication and division	Point	*****, **/**
6	Addition and subtraction	Line	**+**, **-**

Order in the evaluation of equivalent operators

The control evaluates arithmetic operations with equivalent operators from left to right.

Example: 2 + 3 - 2 = (2 + 3) - 2 = 3

Exception: Concatenated powers are evaluated from right to left.

Example: 2 ^ 3 ^ 2 = 2 ^ (3 ^ 2) = 2 ^ 9 = 512

Example: Perform multiplication/division before addition/subtraction

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

- 1st calculation: 5 * 3 = 15
- 2nd calculation: 2 * 10 = 20
- 3rd calculation: 15 + 20 = 35

Example: Calculate power before addition/subtraction

13 Q2 = SQ 10 - 3^3	= 73

- 1st calculation: 10 squared = 100
- 2nd calculation: 3 to the power of 3 = 27
- 3rd calculation: 100 – 27 = 73

Example: Calculate function before power

14 Q4 = SIN 30 ^ 2	= 0.25

- 1st calculation: Calculate sine of 30 = 0.5
- 2nd calculation: 0.5 squared = 0.25

Example: Evaluate expression in parentheses before function

15 Q5 = SIN (50 - 20)	= 0.5

- 1st calculation: Perform operations in parentheses first: 50 – 20 = 30
- 2nd calculation: Calculate sine of 30 = 0.5

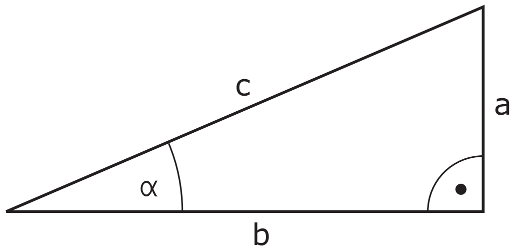
Overview

The control displays the following soft keys:

Soft key	Logical function	Operator
+	**Addition** Example: **Q10 = Q1 + Q5**	Line
-	**Subtraction** Example: **Q25 = Q7 - Q108**	Line
*	**Multiplication** Example: **Q12 = 5 * Q5**	Point
/	**Division** Example: **Q25 = Q1 / Q2**	Point
(	**Open parenthesis** Example: **Q12 = Q1 * (Q2 + Q3)**	Expression in parentheses
)	**Close parenthesis** Example: **Q12 = Q1 * (Q2 + Q3)**	Parentheses
SQ	**Square** (square) Example: **Q15 = SQ 5**	Function
SQRT	**Calculate square root** (square root) Example: **Q22 = SQRT 25**	Function
SIN	**Calculate sine** Example: **Q44 = SIN 45**	Function
COS	**Calculate cosine** Example: **Q45 = COS 45**	Function
TAN	**Calculate tangent** Example: **Q46 = TAN 45**	Function
ASIN	**Calculate arcsine** Inverse function of sine The control determines the angle from the ratio of the opposite side to the hypotenuse. Example: **Q10 = ASIN (Q40 / Q20)**	Function
ACOS	**Calculate arccosine** Inverse function of cosine The control determines the angle from the ratio of the adjacent side to the hypotenuse. Example: **Q11 = ACOS Q40**	Function
ATAN	**Calculate arctangent** Inverse function of tangent The control determines the angle from the ratio of the opposite side to the adjacent side. Example: **Q12 = ATAN Q50**	Function
^	**Exponentiation** Example: **Q15 = 3 ^ 3**	Power

Soft key	Logical function	Operator
PI	**Use the "pi" constant** π = 3.14159 Example: **Q15 = PI**	
LN	**Calculate the natural logarithm (LN)** Base = e = 2.7183 Example: **Q15 = LN Q11**	Function
LOG	**Calculate the logarithm** Base = 10 Example: **Q33 = LOG Q22**	Function
EXP	**Use the exponential function (e ^ n)** Base = e = 2.7183 Example: **Q1 = EXP Q12**	Function
NEG	**Negate** Multiply by –1 Example: **Q2 = NEG Q1**	Function
INT	**Calculate an integer** Truncate decimal places Example: **Q3 = INT Q42** The **INT** function does not round off—it simply truncates the decimal places. **Further information:** "Example: Rounding a value", Page 223	Function
ABS	**Calculate the absolute value** Example: **Q4 = ABS Q22**	Function
FRAC	**Calculate a fraction** Truncate the digits before the decimal point Example: **Q5 = FRAC Q23**	Function
SGN	**Check the algebraic sign** Example: **Q12 = SGN Q50** If **Q50 = 0**, then **SGN Q50 = 0** If **Q50 < 0**, then **SGN Q50 = -1** If **Q50 > 0**, then **SGN Q50 = 1**	Function
%	**Calculate the modulo value (division remainder)** Example: **Q12 = 400 % 360** Result: **Q12 = 40**	Function

Example: Trigonometric function



The lengths of the opposite side a in parameter **Q12** and the adjacent side b in **Q13** are given.

The angle **α** is to be calculated.

Calculate the angle **α** from the opposite side a and the adjacent side b by means of the arc tangent; assign result **Q25**:

Q
- Press the **Q** key

FORMULA
- Press the **FORMULA** soft key
- > The control asks you for the number of the result parameter.
- Enter **25**

ENT
- Press the **ENT** key

▷
- Scroll through the soft-key row

ATAN
- Press the **ATAN** arc tangent function soft key

◁
- Scroll through the soft-key row

(
- Press the **Opening parenthesis** soft key

Q
- Enter **12** (the parameter number)

/
- Select division

Q
- Enter **13** (the parameter number)

)
- Press the **Closing parenthesis** soft key

END
- Press the **END** key to conclude the formula entry

Example

```
37 Q25 = ATAN (Q12/Q13)
```

Example: Rounding a value

The **INT** function truncates the decimal places.

In order for the control to round correctly, rather than simply truncating the decimal places, add the value 0.5 to a positive number. For a negative number you must subtract 0.5.

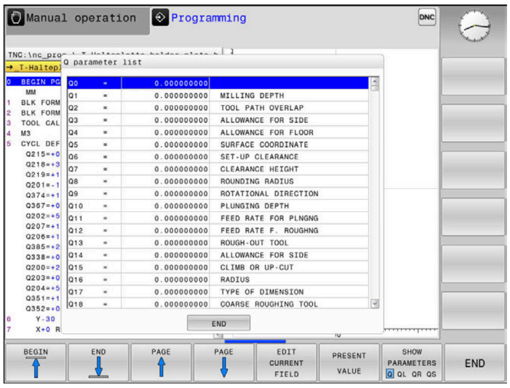
The control uses the **SGN** function to detect whether a number is positive or negative.

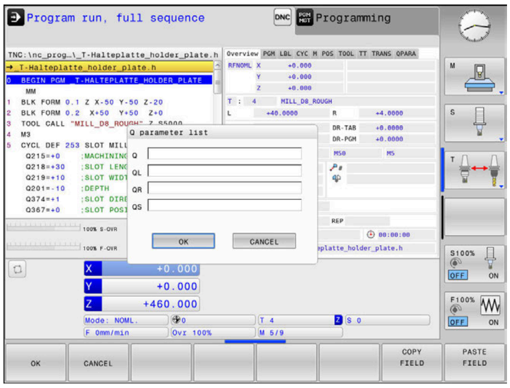
0 BEGIN PGM ROUND MM	
1 FN 0: Q1 = +34.789	First number to be rounded
2 FN 0: Q2 = +34.345	Second number to be rounded
3 FN 0: Q3 = -34.432	Third number to be rounded
4 ;	
5 Q11 = INT (Q1 + 0.5 * SGN Q1)	Add the value 0.5 to Q1, then truncate the decimal places
6 Q12 = INT (Q2 + 0.5 * SGN Q2)	Add the value 0.5 to Q2, then truncate the decimal places
7 Q13 = INT (Q3 + 0.5 * SGN Q3)	Subtract the value 0.5 from Q3, then truncate the decimal places
8 END PGM ROUND MM	

9.8 Checking and changing Q parameters

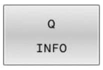
Procedure

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 want to change the value, then press the **EDIT CURRENT FIELD** soft key, enter the new value, and confirm with the **ENT** key
- If you want to leave the value unchanged, then press the **PRESENT VALUE** soft key or close the dialog with the **END** key

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.

While the control is executing an NC program, you cannot edit the variables using the **Q parameter list** window. Changes are only possible while program run has been interrupted or aborted.

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

This status is reached after an NC block has been executed, for example in **Program run, single block**

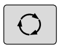
The following Q and QS parameters cannot be edited in the **Q parameter list** window:

- Variable range from 100 to 199, because there might be interferences with special functions in the control.
- Variable range from 1200 to 1399, because there might be interferences with machine manufacturer-specific functions.

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

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

- If needed, interrupt the program run (e.g., by pressing the **NC STOP** key and the **INTERNAL STOP** soft key), or stop the test run



- Display 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 check. 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, where e-08 corresponds to the factor of 10^{-8}.

9.9 Additional functions

Overview

The additional functions appear when the **DIVERSE FUNCTION** soft key is pressed. The control displays the following soft keys:

Soft key	Function	Page
FN14 ERROR=	**FN 14: ERROR** Display error messages	227
FN16 F-PRINT	**FN 16: F-PRINT** Formatted output of texts or Q parameter values	233
FN18 SYS-DATUM READ	**FN 18: SYSREAD** Read system data	243
FN19 PLC=	**FN 19: PLC** Transfer values to the PLC	243
FN20 WAIT FOR	**FN 20: WAIT FOR** NC and PLC synchronization	244
FN26 OPEN TABLE	**FN 26: TABOPEN** Open a freely definable table	297
FN27 WRITE TO TABLE	**FN 27: TABWRITE** Write to a freely definable table	298
FN28 READ FROM TABLE	**FN 28: TABREAD** Read from a freely definable table	300
FN29 PLC LIST=	**FN 29: PLC** Transfer up to eight values to the PLC	245
FN37 EXPORT	**FN 37: EXPORT** Export local Q parameters or QS parameters to a calling NC program	245
FN38 SEND	**FN 38: SEND** Send information from the NC program	246

FN 14: ERROR Output of error messages

With the **FN 14: ERROR** function, you can output error messages under program control. The messages are pre-defined by the machine manufacturer or by HEIDENHAIN.

If, during program run or during simulation, the control executes the **FN 14: ERROR** function, it will interrupt program run and display the defined message. You must then restart the NC program.

Error number range	Error message
0 ... 999	Machine-dependent dialog
1000 ... 2999	Control-dependent dialog
3000 ... 9999	Machine-dependent dialog
10 000 and higher	Control-dependent dialog

Refer to your machine manual.
The machine manufacturer assigns and defines the error numbers up to 999 and from 3000 to 9999.

Example

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

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

The following is a complete list of the **FN 14: ERROR** error messages. Please be aware that not all error messages might be available, depending on the model of your control.

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

Error number	Text
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive 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

Error number	Text
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal 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	3D ROT not permitted
1076	Activate 3D 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
1094	Tool name not permitted

Error number	Text
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
1110	MOVE not possible
1111	Presetting not allowed!
1112	Thread angle too small!
1113	3D ROT status is contradictory!
1114	Configuration is incomplete
1115	No turning tool is active
1116	Tool orientation is inconsistent
1117	Angle not possible!
1118	Radius too small!
1119	Thread runout too short!
1120	Contradictory meas. points
1121	Too many limits
1122	Machining strategy with limits not possible
1123	Machining direction not possible
1124	Check the thread pitch!
1125	Angle cannot be calculated
1126	Eccentric turning not possible
1127	No milling tool is active
1128	Insufficient length of cutting edge
1129	Gear definition is inconsistent or incomplete
1130	No finishing allowance provided
1131	Line does not exist in table
1132	Probing process not possible
1133	Coupling function not possible

Error number	Text
1134	Machining cycle is not supported by this NC software
1135	Touch probe cycle is not supported by this NC software
1136	NC program aborted
1137	Touch probe data incomplete
1138	LAC function not possible
1139	Rounding radius or chamfer is too large!
1140	Axis angle not equal to tilt angle
1141	Character height not defined
1142	Excessive character height
1143	Tolerance error: Workpiece rework
1144	Tolerance error: Workpiece scrap
1145	Faulty dimension definition
1146	Illegal entry in compensation table
1147	Transformation not possible
1148	Tool spindle incorrectly configured
1149	Offset of the turning spindle unknown
1150	Global program settings are active
1151	Faulty configuration of OEM macros
1152	The combination of programmed oversizes is not possible
1153	Measured value not captured
1154	Check the monitoring of the tolerance
1155	Hole is smaller than the stylus tip
1156	Preset cannot be set
1157	Alignment of a rotary table is not possible
1158	Alignment of rotary axes is not possible
1159	Infeed limited to length of cutting edge
1160	Machining depth defined as 0
1161	Tool type is unsuitable
1162	Finishing allowance not defined
1163	Machine datum could not be written
1164	Spindle for synchronization could not be ascertained
1165	Function is not possible in the active operating mode
1166	Oversize defined too large
1167	Number of teeth not defined
1168	Machining depth does not increase monotonously

Error number	Text
1169	Infeed does not decrease monotonously
1170	Tool radius not defined correctly
1171	Mode for retraction to clearance height not possible
1172	Gear wheel definition incorrect
1173	Probing object contains different types of dimension definition
1174	Dimension definition contains impermissible characters
1175	Actual value in dimension definition faulty
1176	Starting point of hole too deep
1177	Dimension def.: Nominal value missing for manual pre-positioning
1178	A replacement tool is not available
1179	OEM macro is not defined
1180	Measurement not possible with auxiliary axis
1181	Start position not possible with modulo axis
1182	Function only possible if door is closed
1183	Number of possible records exceeded
1184	Inconsistent machining plane due to axis angle with basic rotation
1185	Transfer parameter contains an impermissible value
1186	Tooth width RCUTS is defined too large
1187	Usable length LU of the tool is too small
1188	The defined chamfer is too large
1189	Chamfer angle cannot be machined with the active tool
1190	The allowances do not define any stock removal
1191	Spindle angle not unique

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

Fundamentals

With the function **FN 16: F-PRINT**, you can output formatted fixed and variable numbers and texts (e.g., in order to save measuring logs).

You can output the values as follows:

- Save them to a file on the control
- Display them in a window on the screen
- Save them to a file on an external drive or USB device
- Print them to a connected printer

Procedure

In order to output fixed or variable numbers and texts, the following is required:

- Source file
  The source file determines the contents and formatting.
- NC function **FN 16: F-PRINT**
  The control creates the output file using the NC function **FN 16**.
  The maximum size of the output file is 20 kB.

Creating a text file

In order to output formatted text and the values of the Q parameters, use the control's text editor to create a text file. In this file, you can define the format and Q parameters to be output.

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:

> Please note that the input is case-sensitive.

Formatting characters	Meaning
"..."	Identifies the formatting of the contents to be output For text output, you can use the UTF-8 character set.
%F, **%D** or **%I**	Initiate the formatted output of Q, QL and QR parameters ■ **F**: Float (32-bit floating-point number) ■ **D**: Double (64-bit floating-point number) ■ **I**: Integer (32-bit integer)

Formatting characters	Meaning
9.3	Define the number of digits for the output of numerical values ■ 9: Total number of digits, including decimal separator ■ 3: Number of decimal places
%S or **%RS**	Initiate the formatted or unformatted output of a QS parameter ■ **S**: String ■ **RS**: Raw String The control takes over the following text without any changes and formatting.
,	Separate the input within a format-file line (e.g., data type and variable)
;	End of the format-file line
*****	Initiate a comment line within the format file Comments are not included in the output file
%"	Output quotation marks in the output file
%%	Output a percentage sign in the output file
****	Output a backslash in the output file
\n	Output a line break in the output file
+	Output the variable value right-aligned in the output file
-	Output the variable value left-aligned in the output file

Example

Input	Meaning
“X1 = %+9.3 F“, Q31;	Format for the Q parameter: ■ **X1 =**: Output the text **X1 =** ■ **%**: 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	Meaning
CALL_PATH	Output the path name of the NC program that contains the **FN 16** function (e.g., **"TouchProbe: %S",CALL_PATH;**)
M_CLOSE	Close the file written to with **FN 16**
M_APPEND	Upon renewed output, append the contents of the output file to the existing output file
M_APPEND_MAX	Upon renewed output, append the contents of the output file to the existing output file until the maximum file size of 20 kB is reached (e.g., **M_APPEND_MAX20;**)
M_TRUNCATE	Upon renewed output, overwrite the output file
M_EMPTY_HIDE	Do not output blank lines for undefined or empty QS parameters in the output file
M_EMPTY_SHOW	Output blank lines for undefined or empty QS parameters and reset **M_EMPTY_HIDE**
L_ENGLISH	Outputs text only for English conversational language
L_GERMAN	Outputs text only for German conversational language
L_CZECH	Outputs text only for Czech conversational language
L_FRENCH	Outputs text only for French conversational language
L_ITALIAN	Outputs text only for Italian conversational language
L_SPANISH	Outputs text only for Spanish conversational language
L_PORTUGUE	Outputs text only for Portuguese conversational language
L_SWEDISH	Outputs text only for Swedish conversational language
L_DANISH	Outputs text only for Danish conversational language
L_FINNISH	Outputs text only for Finnish conversational language
L_DUTCH	Outputs text only for Dutch conversational language
L_POLISH	Outputs text only for Polish conversational language
L_HUNGARIA	Outputs text only for Hungarian conversational language
L_RUSSIAN	Outputs text only for Russian conversational language

Keyword	Meaning
L_CHINESE	Outputs text only for Chinese conversational language
L_CHINESE_TRAD	Outputs text only for Chinese (traditional) conversational language
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_KOREAN	Outputs text only for Korean 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	Output the hours of the current time
MIN	Output the minutes of the current time
SEC	Output the seconds of the current time
DAY	Output the day of the current date
MONTH	Output the month of the current date
STR_MONTH	Output the month of the current date in short form
YEAR2	Output the year of the current date in two-digit format
YEAR4	Output the year of the current date in four-digit format

Example

Example of a text file to define the output format:

“MEASURING LOG OF IMPELLER CENTER OF GRAVITY“;
“DATE: %02d.%02d.%04d“,DAY,MONTH,YEAR4;
“TIME: %02d:%02d:%02d“,HOUR,MIN,SEC;
“NO. OF MEASURED VALUES: = 1“;
"X1 = %9.3F", Q31;
"Y1 = %9.3F", Q32;
"Z1 = %9.3F", Q33;
L_GERMAN;
"Werkzeuglänge beachten";
L_ENGLISH;
"Remember the tool length";

Example

Example of a format file that generates an output file with variable contents:

“TOUCHPROBE“;
“%S“,QS1;
M_EMPTY_HIDE;
“%S“,QS2;
“%S“,QS3;
M_EMPTY_SHOW;
“%S“,QS4;
M_CLOSE;

Example of an NC program that defines only **QS3**:

11 Q1 = 100	; Assign the value **100** to **Q1**
12 QS3 = "Pos 1: " || TOCHAR(DAT+Q1)	; Convert the numerical value of **Q1** to an alphanumeric value and assign it to the defined string
13 FN 16: F-PRINT TNC:\fn16.a / SCREEN:	; Display the output file with **FN 16** on the control screen

Example of a screen output with two empty lines resulting from **QS1** and **QS4**:

Activating FN 16 output in an NC program

Use the function **FN 16** to define the output file.

The control creates the output file in the following cases:

- End of program **END PGM**
- Cancellation of program with the **NC STOP** key
- **M_CLOSE** keyword in the format file

Enter the path to the text file and the path to 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
- Select the target, i.e. the output path

There are two ways to define the output path:

- Directly in the **FN 16** function
- In the machine parameters, under **CfgUserPath** (no. 102200)

> If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

Specifying the path 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.

As an alternative to complete paths, you can program relative 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 the calling file one folder level up and in another folder **FN 16: F-PRINT ..\MASKE\MASKE1.A/ ..\PROT1.TXT**

Use the **SYNTAX** soft key to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path.

Further information: "File names", Page 97

If the complete path is enclosed in quotation marks, you can use both **** and **/** to separate the folders and files.

> Operating and programming notes:
> - If you define a path both in the machine parameters and in the **FN 16** function, the path in the **FN 16** function has priority.
> - 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 of the previously output 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 type of the output (e.g., TXT, A, XLS, HTML).
> - Use **FN 18** to retrieve 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 243

Defining the output path in machine parameters

If you wish to save the measurement results to a certain directory, you can define the output path for the log file in the machine parameters.

To change the output path:

MOD
- Press the **MOD** key
- Enter the code number 123

↓
- Select the machine parameter **CfgUserPath** (no. 102200)

↓
- Select the machine parameter **fn16DefaultPath** (no. 102202)
- > The control opens a pop-up window.
- Select the output path for the machine operating modes

↓
- Select the machine parameter **fn16DefaultPathSim** (no. 102203)
- > The control opens a pop-up window.
- Select the output path for the **Programming** and **Test Run** operating modes

Enter the source or the target with parameters

You can enter the paths of the source and the output files as variable values. For this purpose, the desired variables must have been defined in the NC program.

Further information: "Assigning string parameters", Page 249

If you want to define variable paths, use the following syntax to enter the QS parameters:

Syntax element	Meaning
:'QS1'	Enter QS parameters with a preceding colon and between single quotation marks
:'QL3'.txt	Specify the file name extension of the target file, if required

> **i** If you want use a QS parameter to output a path 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 PROT1.TXT file:

MEASURING LOG OF IMPELLER CENTER OF GRAVITY

DATE: 15.07.2015

TIME: 08:56:34

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 use the **FN 16** function to display messages in a window on the control screen. This allows you to display explanatory texts in such a way that the user cannot continue without reacting to them. The contents of the output text and the position in the NC program can be chosen freely. You can also output variable values.

In order to display the message on the control screen, enter **SCREEN:** as the output path.

Example

11 FN 16: F-PRINT TNC:\MASKE \MASKE1.A / SCREEN:	; Display the output file with **FN 16** on the control screen

If the message has more lines than can fit in the pop-up window, you can use the arrow keys to scroll through the window.

> If you want to overwrite the previous pop-up window, program the **M_CLOSE** or **M_TRUNCATE** keyword.

Closing the pop-up window

You can close the window in the following ways:

- By pressing the **CE** key
- Defining the **SCLR:** output path (Screen Clear)

Example

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A / SCLR:
```

You can also use the **FN 16: F-PRINT** function to close the pop-up window. In this case, no text file is required.

Example

```
96 FN 16: F-PRINT / SCLR:
```

Exporting messages

With the **FN 16** function, you can save the output files to a drive or a USB device.

To save the output file, define the path including the drive in the **FN 16** function.

Example

11 FN 16: F-PRINT TNC:\MSK-\MSK1.A / PC325:\LOG-\PRO1.TXT	; Save output file with **FN 16**

> If you program the same output multiple times 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 use the **FN 16** function to print output files to a connected printer.

> The connected printer must be PostScript-enabled.
> **Further information:** User's Manual for **Setup, Testing and Running NC Programs**

The control will only print the output file if the source file ends with the **M_CLOSE** keyword.

To use the default printer, enter **Printer:** as the target path and a file name.

If you do not use the default printer, enter the path to the respective printer (e.g., **Printer:\PR0739**) and a file name.

The control saves the file using the defined file name and the defined path. The control will not print the file name.

The control saves the file temporarily until printing is complete.

Example

11 FN 16: F-PRINT TNC:\MASKE-\MASKE1.A / PRINTER:-\PRINT1	; Print output file with **FN 16**

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.
>
> As an alternative, you can use **TABDATA READ** to read out data from the active tool table. In this case, the control will automatically convert the table values to the unit of measure used in the NC program.

Further information: "System data", Page 518

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 Transferring values to PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers to communicate with the PLC from within an NC program. It is not recommended that machine operators or NC programmers use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

The **FN 19: PLC** function transfers up to two fixed or variable values 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., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers to communicate with the PLC from within an NC program. It is not recommended that machine operators or NC programmers use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

With the **FN 20: WAIT FOR** function, you can synchronize the NC and the PLC during program run. The control stops program run until the condition you specified in the **FN 20: WAIT FOR-** block has been met.

The **SYNC** function is used whenever you read system data (e.g., with **FN 18: SYSREAD**). The system data need to be synchronized with the current date and time. Use the **FN 20: WAIT FOR** to stop the look-ahead calculation. When the control encounters **FN 20**, it will only calculate the NC block after it has executed the NC block that contains **FN 20**.

Example: Pause internal look-ahead calculation, read current position in the X axis

11 FN 20: WAIT FOR SYNC	; Stop internal look-ahead calculation with **FN 20**
12 FN 18: SYSREAD Q1 = ID270 NR1 IDX1	; Determine the position of the X axis with **FN 18**

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., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers to communicate with the PLC from within an NC program. It is not recommended that machine operators or NC programmers use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

The **FN 29: PLC** function transfers up to eight fixed or variable values to the PLC.

FN 37: EXPORT

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers to communicate with the PLC from within an NC program. It is not recommended that machine operators or NC programmers use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

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

FN 38: SEND – Sending information from the NC program

The function **FN 38: SEND** enables you to retrieve fixed or variable values from the NC program and write them to the log or send them to an external application (e.g., StateMonitor).

The syntax consists of two parts:

- **Format of transmitted text**: Output text with optional placeholders for variable values (e.g., **%f**)

  > Input may be in the form of QS parameters.
  >
  > Both fixed and variable numbers and texts are case-sensitive, so enter them correctly.

- **Datum for placeholder in text**: List of max. seven Q, QL, or QR variables (e.g., **Q1**)

Data transmission is through a standard TCP/IP computer network.

> For more detailed information, consult the RemoTools SDK manual.

Example

Document the values from **Q1** and **Q23** in the log.

```
FN 38: SEND /"Q-Parameter Q1: %f Q23: %f" / +Q1 / +Q23
```

Example

Define the output format for the variable values.

```
FN 38: SEND /"Q-Parameter Q1: %05.1f" / +Q1
```

> The control outputs the variable value as a five-digit number, of which one digit is a decimal place. The output will be padded with leading zeroes as needed.

```
FN 38: SEND /"Q-Parameter Q1: % 1.3f" / +Q1
```

> The control outputs the variable value as a seven-digit number, of which three digits are decimal places. The output will be padded with blank spaces as needed.

> To obtain **%** in the output text, enter **%%** at the desired position.

Example

In this example, you will send information to StateMonitor.

With the function **FN 38**, you can, for example, enter job data.

The following requirements must be met in order to use this function:

- StateMonitor version 1.2
  Job management with JobTerminal (option 4) is possible with StateMonitor version 1.2 or higher
- The job has been entered in StateMonitor
- Machine tool has been assigned

The following stipulations apply to this example:

- Job number 1234
- Working step 1

FN 38: SEND /"JOB:1234_STEP:1_CREATE"	Create job
FN 38: SEND /"JOB:1234_STEP:1_CREATE_ITEMNAME: HOLDER_ITEMID:123_TARGETQ:20"	Alternative: Create job with part name, part number, and required quantity
FN 38: SEND /"JOB:1234_STEP:1_START"	Start job
FN 38: SEND /"JOB:1234_STEP:1_PREPARATION"	Start preparation
FN 38: SEND /"JOB:1234_STEP:1_PRODUCTION"	Production
FN 38: SEND /"JOB:1234_STEP:1_STOP"	Stop job
FN 38: SEND /"JOB:1234_STEP:1_ FINISH"	Finish job

You can also report the quantity of workpieces of the job.

With the **OK**, **S**, and **R** placeholders, you can specify whether the quantity of reported workpieces has been machined correctly or not.

With **A** and **I** you define how StateMonitor interprets the response. If you transfer absolute values, StateMonitor overwrites the previously valid values. If you transfer incremental values, StateMonitor increments the quantity.

FN 38: SEND /"JOB:1234_STEP:1_OK_A:23"	Actual quantity (OK) absolute
FN 38: SEND /"JOB:1234_STEP:1_OK_I:1"	Actual quantity (OK) incremental
FN 38: SEND /"JOB:1234_STEP:1_S_A:12"	Scrap (S) absolute
FN 38: SEND /"JOB:1234_STEP:1_S_I:1"	Scrap (S) incremental
FN 38: SEND /"JOB:1234_STEP:1_R_A:15"	Rework (R) absolute
FN 38: SEND /"JOB:1234_STEP:1_R_I:1"	Rework (R) incremental

9.10 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 202

The **STRING FORMULA** and **FORMULA** Q parameter functions contain various functions for processing the string parameters.

Soft key	Functions of the STRING FORMULA	Page
DECLARE STRING	Assigning string parameters	249
CFGREAD	Read out the machine parameter values	258
STRING FORMULA	Chain-linking string parameters	250
TOCHAR	Converting a numerical value to a string parameter	251
SUBSTR	Copy a substring from a string parameter	252
SYSSTR	Read system data	253

Soft key	Formula string functions	Page
TONUMB	Converting a string parameter to a numerical value	254
INSTR	Checking a string parameter	255
STRLEN	Finding the length of a string parameter	256
STRCOMP	Compare alphabetic priority	257

If you use the **STRING FORMULA** function, the result is always an alphanumeric value. If you use the **FORMULA** function, the result is always a numeric value.

Assigning 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

11 DECLARE STRING QS10 = "workpiece"	; Assign alphanumeric value to **QS10**

Chain-linking string parameters

With the concatenation operator (string parameter **||** string parameter) you can make a chain of two or more string parameters.

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

STRING FUNCTIONS

- Press the **STRING FUNCTIONS** soft key

STRING FORMULA

- Press the **STRING FORMULA** soft key

ENT

- Enter the number of the string parameter in which the control is to save the concatenated string. Confirm with the **ENT** key.
- Enter the number of the string parameter in which the **first** substring is saved. Confirm with the **ENT** key
- > The control shows the concatenation symbol **||**
- 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 and QS13

11 QS10 = QS12 || QS13	; Concatenate contents of **QS12** and **QS13** and assign them to the QS parameter **QS10**

Parameter contents:

- **QS12**: **Status:**
- **QS13**: **Scrap**
- **QS10**: **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

11 QS11 = TOCHAR (DAT+Q50 DECIMALS3)	; Convert a numerical value from **Q50** to an alphanumeric value and assign it to the QS parameter **QS11**

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 copying 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)

11 QS13 = SUBSTR (SRC_QS10 BEG2 LEN4)	; Assign substring from **QS10** to the QS parameter **QS13**

Reading system data

With the **SYSSTR** NC function, you can read system data and save the contents in QS parameters. Select the system datum by means of a group number **(ID)** and a number **(NR)**.

Optionally, you can enter **IDX** and **DAT**.

Group name, ID no.	Number	Meaning
Program information, 10010	1	Path of the current main program or pallet program
	2	Path of the currently executed NC program
	3	Path of the NC program selected with Cycle **12 PGM CALL**
	10	Path of the NC program selected with **SEL PGM**
Channel data, 10025	1	Name of the current channel (e.g., **CH_NC**)
Values programmed in the tool call, 10060	1	Current tool name The NC function saves the tool name only if the tool has been called using its tool name.
Current system time, 10321	1 to 16, 20	■ 1: D.MM.YYYY h:mm:ss ■ 2: D.MM.YYYY h:mm ■ 3: D.MM.YY hh:mm ■ 4: YYYY-MM-DD hh:mm:ss ■ 5: YYYY-MM-DD hh:mm ■ 6: YYYY-MM-DD h:mm ■ 7: YY-MM-DD h:mm ■ 8: DD.MM.YYYY ■ 9: D.MM.YYYY ■ 10: D.MM.YY ■ 11: YYYY-MM-DD ■ 12: YY-MM-DD ■ 13: hh:mm:ss ■ 14: h:mm:ss ■ 15: h:mm ■ 16: DD.MM.YYYY hh:mm ■ 20: XX "XX" stands for the two-digit number of the current calendar week that—in accordance with ISO 8601 —is characterized by the following: ■ It comprises seven days ■ It begins with Monday ■ It is numbered sequentially ■ The first calendar week (week 01) is the week with the first Thursday of the Gregorian year.
Touch-probe data, 10350	50	Type of the active TS workpiece touch probe
	70	Type of the active TT tool touch probe
	73	Name of the active TT workpiece touch probe from the **activeTT** machine parameter

Group name, ID no.	Number	Meaning
	2	Path of the currently selected pallet table
NC software version, 10630	10	Number of the NC software version
Tool data, 10950	1	Current tool name
	2	Content of the **DOC** column of the current tool
	4	Tool-carrier kinematics of the current tool

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 function

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

11 Q82 = TONUMB (SRC_QS11)	; Convert alphanumeric value from **QS11** to a numerical value and assign it to **Q82**

Testing a string parameter

The **INSTR** function checks whether (and where) a string parameter is contained in another string parameter.

Q

- Select Q parameter function

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)	; Search **QS10** for substring from **QS13**

Determining 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 determined string length, and confirm with the **ENT** key

◁

- Shift the soft-key row

STRLEN

- Select the function for finding the text length of a string parameter
- Enter the number of the QS parameter whose length is to be determined, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your input with the **END** key

Example: Find the length of QS15

11 Q52 = STRLEN (SRC_QS15)	; Determine the number of characters in **QS15** and assign it to **Q52**

> If the selected QS parameter has not been defined, the control returns the value **-1**.

Comparing the lexical order of two alphanumerical strings

With the **STRCOMP** NC function, you can compare the lexical order of the content of two QS parameters.

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 input with the **END** key

The control returns the following results:

- **0**: The content of the two parameters is identical
- **-1**: In the lexical order, the content of the first QS parameter comes **before** the content of the second QS parameter
- **+1**: In the lexical order, the content of the first QS parameter comes **after** the content of the second QS parameter

The lexical order is as follows:

1 Special characters (e.g., ?_)
2 Numerals (e.g., 123)
3 Uppercase letters (e.g., ABC)
4 Lowercase letters (e.g., abc)

Starting from the first character, the control proceeds until the contents of the QS parameters differ from each other. If the contents differ starting from, for example, the fourth digit, the control aborts the check at this point.

Shorter contents with identical strings are displayed first in the order (e.g., abc before abcd).

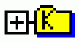
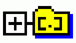
Example: Compare the lexical order of QS12 and QS14

11 Q52 = STRCOMP (SRC_QS12 SEA_QS14)	; Compare the lexical order of the values of **QS12** and **QS14**

Reading out machine parameters

With the **CFGREAD** NC function, you can read out machine parameter contents of the control as numerical or alphanumeric values. The read-out numerical values are always given in metric form.

To read a machine parameter, you need to determine the following contents in the configuration editor of the control:

Icon	Type	Meaning	Example
	**Key**	Group name of the machine parameter The group name can be specified optionally	CH_NC
	**Entity**	Parameter object The name always begins with **Cfg**	**CfgGeoCycle**
	Attribute	Name of the machine parameter	**displaySpindleErr**
	**Index**	List index of the machine parameter The list index can be specified optionally	[0]

> You can change the display of the existing parameters in the configuration editor for the machine parameter. By default, 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 read out a machine parameter with the **CFGREAD** NC function, you must first define a QS parameter with attribute, entity and key.

The control queries the following parameters in the **CFGREAD** NC function:

- **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 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 **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

11 QS11 = "CH_NC"	; Assign the key to the QS parameter **QS11**
12 QS12 = "CfgGeoCycle"	; Assign the entity to the QS parameter **QS12**
13 QS13 = "pocketOverlap"	; Assign the attribute to the QS parameter **QS13**
14 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	Read out the contents of the machine parameter

9.11 Preassigned Q parameters

For example, the control assigns the following values to the Q parameters **Q100** to **Q199**:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Measurement results from touch-probe cycles

The control saves the values of the Q parameters **Q108** and **Q114** 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 manufacturer, and suppliers.
- Check the machining sequence using a graphic simulation

Preassigned variables, such as Q and QS parameters in the range of 100 to 199, must not be used as calculated parameters in NC programs.

Values from the PLC: Q100 to Q107

The control assigns values from the PLC to the Q parameters **Q100** to **Q107**.

Active tool radius: Q108

The control assigns the value of the active tool radius to the Q parameter **Q108**.

The active tool radius is calculated from the following values:

- Tool radius **R** from the tool table
- Delta value **DR** from the tool table
- Delta value **DR** from the NC program, if a compensation table or tool call is used

Further information: "Delta values for lengths and radii", Page 119

The control will remember the active tool radius even after a restart of the control.

Tool axis: Q109

The value of the Q parameter **Q109** depends on the current tool axis:

Q parameters	Tool axis
Q109 = –1	No tool axis defined
Q109 = 0	X axis
Q109 = 1	Y axis
Q109 = 2	Z axis
Q109 = 6	U axis
Q109 = 7	V axis
Q109 = 8	W axis

Spindle status: Q110

The value of the Q parameter **Q110** depends on the M function last activated for the spindle:

Q parameters	M function
Q110 = –1	No spindle status defined
Q110 = 0	**M3** Switch spindle on clockwise
Q110 = 1	**M4** Switch spindle on counterclockwise
Q110 = 2	**M5** after **M3** Stop the spindle
Q110 = 3	**M5** after **M4** Stop the spindle

Coolant on/off: Q111

The value of the Q parameter **Q111** depends on the M function for the coolant on/off function that was last activated:

Q parameters	M function
Q111 = 1	**M8** Switch coolant supply on
Q111 = 0	**M9** Switch coolant supply off

Overlap factor: Q112

The control assigns the overlap factor for pocket milling to the Q parameter **Q112**.

Unit of measure in the NC program Q113

The value of the Q parameter **Q113** depends on the unit of measure selected in the NC program. In case of program nesting (e.g., with **CALL PGM**), the control will use the unit of measure defined for the main program:

Q parameters	Unit of measure of the main program
Q113 = 0	Metric system (mm)
Q113 = 1	Imperial system (inch)

Tool length: Q114

The control assigns the value of the active tool length to the Q parameter **Q114**.

The active tool length is calculated from the following values:

- Tool length **L** from the tool table
- Delta value **DL** from the tool table
- Delta value **DL** from the NC program, if a compensation table or tool call is used

> The control remembers the active tool length even after a restart of the control.

Measurement result from programmable touch-probe cycles: Q115 to Q119

The control assigns the measurement result of a programmable touch-probe cycle to the following Q parameters.

For these Q parameters, the control does not take the radius and length of the stylus into account.

> The help graphics of the touch-probe cycles show whether the control saves a measurement result in a variable or not.

The control assigns the coordinate axis values after probing to the Q parameters **Q115** to **Q119**:

Q parameters	Axis coordinates
Q115	**TOUCH POINT IN X**
Q116	**TOUCH POINT IN Y**
Q117	**TOUCH POINT IN Z**
Q118	**TOUCH POINT 4TH AXIS** (e.g., A axis) The machine manufacturer defines the 4th axis
Q119	**TOUCH POINT 5TH AXIS** (e.g., B axis) The machine manufacturer defines the 5th axis

Q parameters Q115 and Q116 for automatic tool measurement

The control assigns the deviation of the actual value from the nominal value in automatic tool measurements (e.g., with a TT 160) to the Q parameters **Q115** and **Q116**:

Q parameters	Deviation of actual from nominal value
Q115	Tool length
Q116	Tool radius

After probing, the Q parameters **Q115** and **Q116** might contain other values.

9.12 Accessing tables with SQL statements

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 available SQL commands.

The syntax of the SQL commands available on the control is strongly influenced by the SQL programming language but does not conform with it entirely. In addition, the control does not support the full 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 data are input or read.

> Read- and write-accesses to individual values in a table can likewise be carried out using the functions **FN 26: TABOPEN**, **FN 27: TABWRITE**, and **FN 28: TABREAD**.
> **Further information:** "Freely definable tables", Page 294
>
> HEIDENHAIN recommends that you use SQL functions instead of **FN 26**, **FN 27**, or **FN 28** in order to achieve maximum HDR hard-disk speeds for table applications and to reduce the amount of computing power used.

The following terms will be used (along with others) below:

- "SQL command" refers to the available soft keys
- "SQL instructions" describe miscellaneous functions that are entered manually as part of the syntax
- A **HANDLE** in the syntax identifies a certain transaction (followed by the parameter for identification)
- A **result set** contains the result of the query

SQL transaction

In the NC software, table accesses occur through an SQL server. This server is controlled via the available SQL commands. The SQL commands can be defined directly in an NC program.

The server is based on a transaction model. A **transaction** consists of multiple steps that are executed together, thereby ensuring that the table entries are processed in an orderly and well-defined manner.

Example of 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**, or **SQL INSERT**
- Confirm or discard interaction using **SQL COMMIT** or **SQL ROLLBACK**
- Approve bindings between table columns and Q parameters using **SQL BIND**

> You must conclude all transactions that have been started—even exclusively reading 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.

Result set and handle

The **result set** contains a subset of a table file. It results from a **SELECT** query performed on the table.

The **result set** is created when a query is executed in the SQL server, thereby occupying resources there.

This query has the same effect as applying a filter to the table, so that only part of the data records become visible. To perform this query, the table file must be read at this point.

The SQL server assigns a **handle** to the **result set**, which enables you to identify the result set for reading or 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**, i.e. it was not possible to create a **result set** for that query. If no rows are found that satisfy the specified condition, 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** or **Positioning with MDI** operating modes:

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

NOTICE

Danger of collision!

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, you save a length from a table to a Q parameter, then the value is thereafter always in metric units. If this value is then used for the purpose of positioning in an inch program (**L X +Q1800**), then an incorrect position will result.

- In inch programs, convert the read value prior to use

NOTICE

Danger of collision!

If you simulate an NC program that includes SQL commands, the control might overwrite table values. Overwriting table values might result in incorrect positioning of the machine. There is a danger of collision.

- Program NC programs in such a way that SQL commands are not executed during simulation
- Use **FN18: SYSREAD ID992 NR16** to check whether the NC program is active in a different operating mode or in **Simulation**

Overview of functions

Overview of soft keys

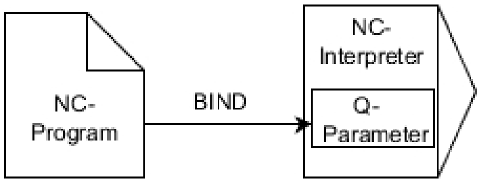
The control offers the following ways of working with SQL commands:

Soft key	Function	Page
SQL BIND	**SQL BIND** creates or disconnects a binding between table columns and Q or QS parameters	268
SQL EXECUTE	**SQL EXECUTE** opens a transaction for selected table columns and table rows or enables the use of other SQL instructions (miscellaneous functions).	269
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	274
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	280
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	279
SQL UPDATE	**SQL UPDATE** expands the transaction to include the change of an existing row	276
SQL INSERT	**SQL INSERT** creates a new table row	278
SQL SELECT	**SQL SELECT** reads out a single value from a table and does not open any transaction	282

SQL BIND

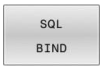
SQL BIND links a Q parameter to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) during data transfer between the **result set** and the NC program.

An **SQL BIND** command without a table name or column name cancels the binding. At the latest, the binding is terminated at the end of the NC program or subprogram.



Programming notes:

- Program any number of bindings with **SQL BIND...**, before using the **FETCH**, **UPDATE**, or **INSERT** commands.
- During the read and write operations, the control considers only those columns that you have specified by means of the **SELECT** command. If you specify columns without a binding in the **SELECT** command, then the control interrupts the read or write operation with an error message.



- **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

Example: Binding Q parameters to table columns

11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	

Example: Remove binding

91 SQL BIND Q881	
92 SQL BIND Q882	
93 SQL BIND Q883	
94 SQL BIND Q884	

SQL EXECUTE

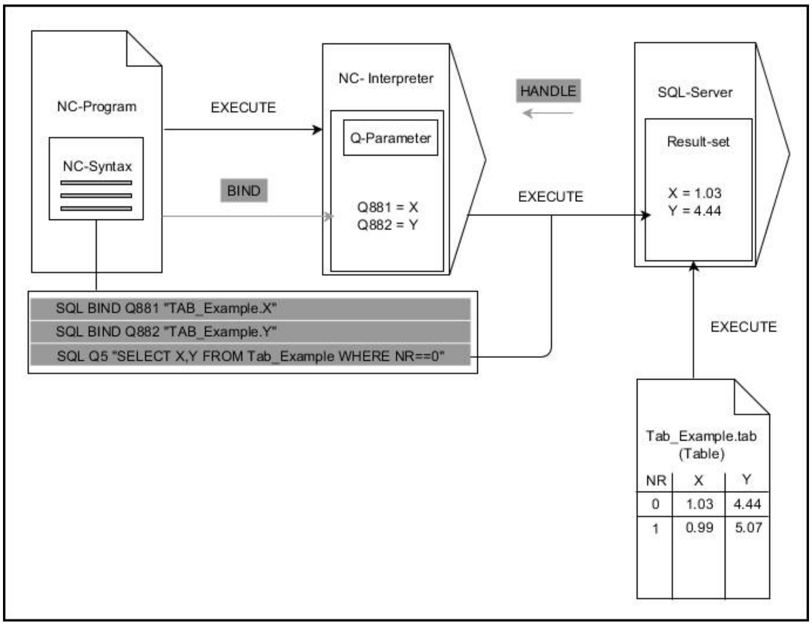
SQL EXECUTE can be used in conjunction with various SQL instructions.

The following SQL instructions are used in the SQL command **SQL EXECUTE**.

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate table
COPY TABLE	Copy table
RENAME TABLE	Rename table
DROP TABLE	Delete table
INSERT	Insert table rows
UPDATE	Update table rows
DELETE	Delete table rows
ALTER TABLE	■ Add table columns using **ADD** ■ Delete table columns using **DROP**
RENAME COLUMN	Rename table columns

> If you use the **SQL EXECUTE** NC function, the control will insert the **SQL** syntax element into the NC program only.

Example for the SQL EXECUTE command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL EXECUTE** command
- Black arrows and associated syntax indicate internal processes of **SQL EXECUTE**

SQL EXECUTE with the SQL SELECT instruction

The SQL server places the data in the **result set** row-by-row. The rows are numbered in ascending order, starting with 0. The SQL commands **FETCH** and **UPDATE** use these row numbers (the **INDEX**).

SQL EXECUTE, in conjunction with the SQL instruction **SELECT**, selects the table values, transfers them to the **result set**, and always opens a transaction in the process. Unlike the SQL command **SQL SELECT**, the combination of **SQL EXECUTE** and the **SELECT** instruction allows multiple columns and rows to be selected at the same time.

Enter the search criteria in the **SQL ... "SELECT...WHERE..."** function. You thereby 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.

Enter the ordering criteria in the **SQL ... "SELECT...ORDER BY..."** function. 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 meet the search criterion, then the SQL server returns a valid **HANDLE** without table entries.

SQL EXECUTE

- Define **Parameter number for result**
  - The return value serves as an identifying feature of a successfully opened transaction
  - The return value is used to control the read operation

    In the specified parameters, the control stores the **HANDLE** under which the read operation will subsequently occur. The **HANDLE** is valid until you confirm or reject the transaction.
  - **0**: Faulty read operation
  - Unequal to **0**: Return value of the **HANDLE**
- **Database: SQL instruction**: Program an SQL instruction
  - **SELECT**: Table columns to be transferred (separate multiple columns with **,**)
  - **FROM**: Synonym or absolute path of the table (path in single quotation marks)
  - **WHERE** (optional): Column names, condition, and comparison value (Q parameters after **:** in single quotation marks)
  - **ORDER BY** (optional): Column names and type of ordering (**ASC** for ascending and **DESC** for descending order)
  - **FOR UPDATE** (optional): To lock other processes from performing a write access to the selected rows

Conditions for WHERE entires

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

Example: selecting table rows

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
. . .
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example"
```

Example: selecting table rows with the WHERE function

```
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example WHERE
   Position_Nr<20"
```

Example: selecting table rows with the WHERE function and Q parameter

```
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example WHERE
   Position_Nr==:'Q11'"
```

Example: defining the table name with absolute path information

```
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM 'V:\table\Tab_Example' WHERE
   Position_Nr<20"
```

Example: generating a table with CREATE TABLE

```
0  BEGIN PGM SQL_CREATE_TAB MM
1  SQL Q10 "CREATE SYNONYM NEW FOR 'TNC:\table     ; Create synonym
   \NewTab.TAB'"
2  SQL Q10 "CREATE TABLE NEW AS SELECT X,Y,Z FROM  ; Create table
   'TNC:\prototype_for_NewTab.tab'"
3  END PGM SQL_CREATE_TAB MM
```

> The sequence of the columns in the created file corresponds to the sequence within the **AS SELECT** instruction.
>
> You can also define synonyms for tables that have not yet been generated.

Example: generating a table with CREATE TABLE and QS

> - If you check the content of a QS parameter in the additional status indicator (**QPARA** tab), then you will see only the first 30 characters and therefore not the entire content.
> - For the instructions within the SQL command, you can likewise use single or combined QS parameters.
> - After the **WHERE** syntax element, you can define the comparison value, which can also be a variable. If you use Q, QL, or QR parameters for the comparison, the control will round the defined value to the next integer. If you use a QS parameter, the control will use the exact value you specified.

```
0 BEGIN PGM SQL_CREATE_TABLE_QS MM
1 DECLARE STRING QS1 = "CREATE TABLE "
2 DECLARE STRING QS2 = "'TNC:\nc_prog\demo\Doku
  \NewTab.t' "
3 DECLARE STRING QS3 = "AS SELECT "
4 DECLARE STRING QS4 = "DL,R,DR,L "
5 DECLARE STRING QS5 = "FROM "
6 DECLARE STRING QS6 = "'TNC:\table\tool.t'"
7 QS7 = QS1 || QS2 || QS3 || QS4 || QS5 || QS6
8 SQL Q1800 QS7
9 END PGM SQL_CREATE_TABLE_QS MM
```

Examples

The following examples do not result in a cohesive NC program. The NC blocks show only possible uses of the SQL command **SQL EXECUTE**.

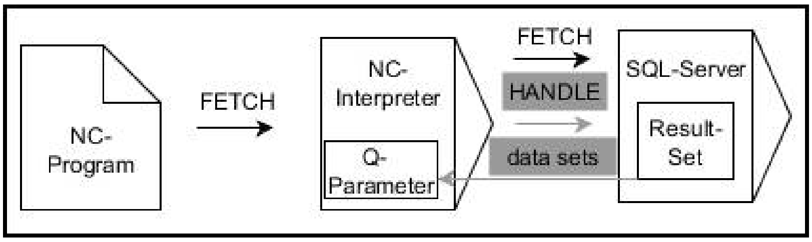
9 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:-\table\WMAT.TAB'"	Create synonym
9 SQL Q1800 "DROP SYNONYM my_table"	Delete synonym
9 SQL Q1800 "CREATE TABLE my_table (NR,WMAT)"	Create table with the columns NR and WMAT
9 SQL Q1800 "COPY TABLE my_table TO 'TNC:\table-\WMAT2.TAB'"	Copy table
9 SQL Q1800 "RENAME TABLE my_table TO 'TNC:\table-\WMAT3.TAB'"	Rename table
9 SQL Q1800 "DROP TABLE my_table"	Delete table
9 SQL Q1800 "INSERT INTO my_table VALUES (1,'ENAW',240)"	Insert table row
9 SQL Q1800 "DELETE FROM my_table WHERE NR==3"	Delete table row
9 SQL Q1800 "ALTER TABLE my_table ADD (WMAT2)"	Insert table column
9 SQL Q1800 "ALTER TABLE my_table DROP (WMAT2)"	Delete table column
9 SQL Q1800 "RENAME COLUMN my_table (WMAT2) TO (WMAT3)"	Rename table column

SQL FETCH

SQL FETCH reads a row from the **result set**. The values of the individual cells are stored by the control in the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**.

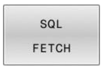
SQL FETCH takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

Example for the SQL FETCH command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL FETCH** command
- Black arrows and associated syntax indicate internal processes of **SQL FETCH**



- Define **Parameter number for result** (return values for the control):
  - **0**: Successful read operation
  - **1**: Faulty read operation
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)
- Define **Database: Index for SQL result** (row number within the **result set**)
  - Row number
  - Q parameter with the index
  - None defined: access to row 0

> The optional syntax elements **IGNORE UNBOUND** and **UNDEFINE MISSING** are intended for the machine manufacturer.

Example: Transfer row number in the Q parameter

11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	
. . .	
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"	
. . .	
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2	

Example: Program the row number directly

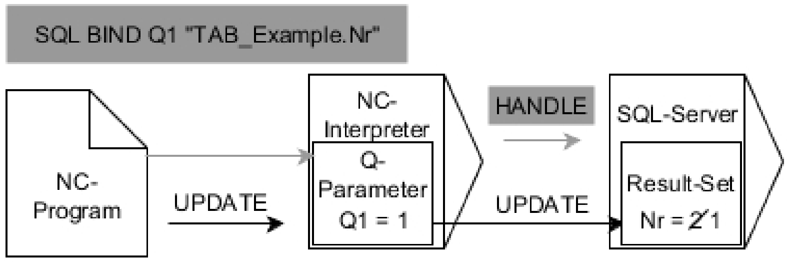
30 SQL FETCH Q1 HANDLE Q5 INDEX5	

SQL UPDATE

SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**. The control completely overwrites the already existing rows in the **result set**.

SQL UPDATE takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

Example for the SQL UPDATE command



The gray arrows and associated syntax do not directly belong to the **SQL UPDATE**
Black arrows and associated syntax show internal processes of **SQL UPDATE**

- Define **Parameter number for result** (return values for the control):
  - **0**: Change was successful
  - **1**: Change failed
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)
- Define **Database: Index for SQL result** (row number within the **result set**)
  - Row number
  - Q parameter with the index
  - None defined: access to row 0

> When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example: Transfer row number in the Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.Position_NR"
12 SQL BIND Q882 "TAB_EXAMPLE.Measure_X"
13 SQL BIND Q883 "TAB_EXAMPLE.Measure_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.Measure_Z"
. . .
20 SQL Q5 "SELECT
   Position_NR,Measure_X,Measure_Y,Measure_Z FROM
   TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: Program the row number directly

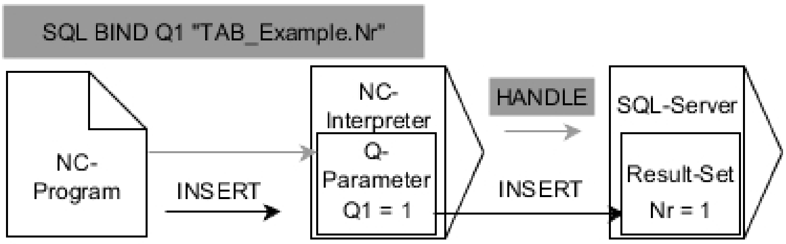
```
40 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

SQL INSERT

SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified.

SQL INSERT takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**). Table columns without a corresponding **SELECT** instruction (not contained in the query result) are described by the control with default values.

Example for the SQL INSERT command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL INSERT** command
- Black arrows and associated syntax indicate internal processes of **SQL INSERT**

SQL INSERT

- Define **Parameter number for result** (return values for the control):
  - **0**: Transaction successful
  - **1**: Transaction failed
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)

> When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example: Transfer row number in the Q parameter

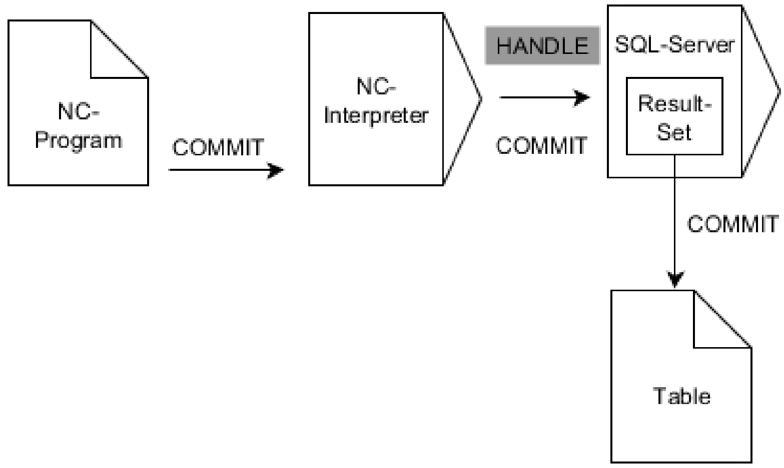
11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	
. . .	
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"	
. . .	
40 SQL INSERT Q1 HANDLE Q5	

SQL COMMIT

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. In this context, a lock that has been set with **SELECT...FOR UPDATE** resets the control.

The assigned **HANDLE** (operation) loses its validity.

Example for the SQL COMMIT command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL COMMIT** command
- Black arrows and associated syntax indicate internal processes of **SQL COMMIT**

SQL COMMIT

- Define **Parameter number for result** (return values for the control):
  - **0**: Transaction successful
  - **1**: Transaction failed
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)

Example

11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	
. . .	
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_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	

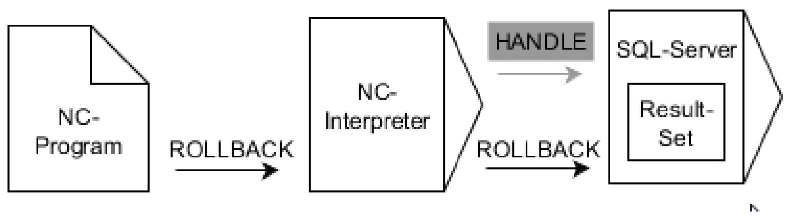
SQL ROLLBACK

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**:

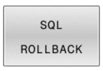
- Without **INDEX**:
  - The control discards all changes and additions of the transaction
  - The control resets a lock set with **SELECT...FOR UPDATE**
  - The control completes the transaction (the **HANDLE** loses its validity)
- With **INDEX**:
  - Only the indexed row remains in the **result set** (the control removes all of the other rows)
  - The control discards any changes and additions that may have been made in the non-specified rows
  - The control locks only those rows indexed with **SELECT...FOR UPDATE** (the control resets all of the other locks)
  - The specified (indexed) row is then the new Row 0 of the **result set**
  - The control does **not** complete the transaction (the **HANDLE** keeps its validity)
  - The transaction must be completed manually with **SQL ROLLBACK** or **SQL COMMIT** at a later time

Example for the SQL ROLLBACK command



Remarks:

- The gray arrows and associated syntax do not directly belong to the **SQL ROLLBACK** command
- Black arrows and associated syntax indicate internal processes of **SQL ROLLBACK**



- Define **Parameter number for result** (return values for the control):
  - **0**: Transaction successful
  - **1**: Transaction failed
- **Database: SQL access ID**: Define Q parameter for the **HANDLE** (for identifying the transaction)
- Define **Database: Index for SQL result** (row that remains in the **result set**)
  - Row number
  - Q parameter with the index

Example

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
. . .
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,
   Measure_Z FROM Tab_Example"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL ROLLBACK Q1 HANDLE Q5
```

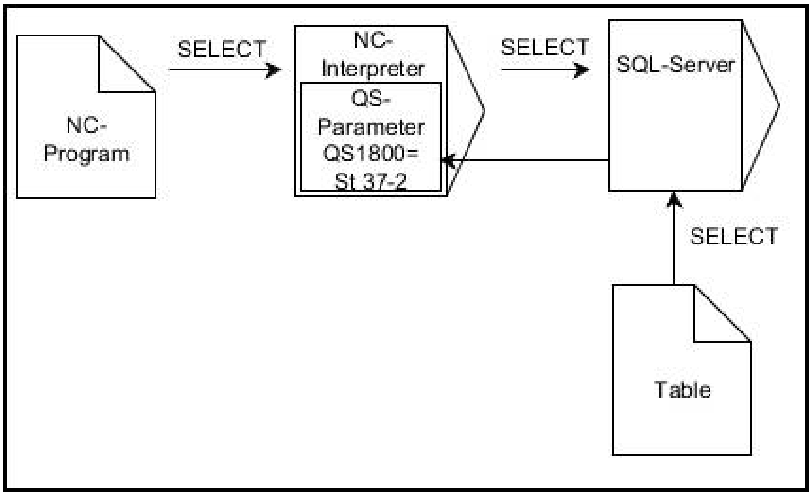
SQL SELECT

SQL SELECT reads a single value from a table and saves the result in the defined Q parameter.

> You can select multiple values or multiple columns using the SQL command **SQL EXECUTE** and the **SELECT** instruction.
> **Further information:** "SQL EXECUTE", Page 269

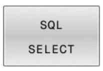
With **SQL SELECT**, there is neither a transaction nor a binding between the table column and Q parameter. The control does not consider any bindings that may exist to the specified column. The control copies the read value only into the parameter specified for the result.

Example for the SQL SELECT command



Remark:

- Black arrows and associated syntax show internal processes of **SQL SELECT**



- Define **Parameter number for result** (Q parameter for saving the value)
- **Database: SQL command text**: Program the SQL instruction
  - **SELECT**: Table column of the value to be transferred
  - **FROM**: Synonym or absolute path of the table (path in single quotation marks)
  - **WHERE**: Column designation, condition, and comparison value (Q parameter after **:** in single quotation marks)

Example: Read and save a value

```
20 SQL SELECT Q5 "SELECT Mess_X FROM Tab_Example
   WHERE Position_NR==3"
```

Comparison

The results of the following NC programs are identical.

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\WMAT.TAB'"	Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NR==3"	Define search
...	

...	
3 SQL SELECT QS1800 "SELECT WMAT FROM my_table WHERE NR==3"	Read and save a value
...	

- If you check the content of a QS parameter in the additional status indicator (**QPARA** tab), then you will see only the first 30 characters and therefore not the entire content.
- For the instructions within the SQL command, you can likewise use single or combined QS parameters.
- After the **WHERE** syntax element, you can define the comparison value, which can also be a variable. If you use Q, QL, or QR parameters for the comparison, the control will round the defined value to the next integer. If you use a QS parameter, the control will use the exact value you specified.

...	
3 DECLARE STRING QS1 = "SELECT "	
4 DECLARE STRING QS2 = "WMAT "	
5 DECLARE STRING QS3 = "FROM "	
6 DECLARE STRING QS4 = "my_table "	
7 DECLARE STRING QS5 = "WHERE "	
8 DECLARE STRING QS6 = "NR==3"	
9 QS7 = QS1 || QS2 || QS3 || QS4 || QS5 || QS6	
10 SQL SELECT QL1 QS7	
11 ...	

Examples

In the following example, the defined material is read from the table (**WMAT.TAB**) and is stored as a text in a QS parameter. The following example shows a possible application and the necessary program steps.

> You can use the **FN 16** function, for example, in order to reuse QS parameters in your own log files.
> **Further information:** "Fundamentals", Page 233

Example: Use a synonym

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table\WMAT.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_READ_WMAT MM	

Step	Explanation
1 Create synonym	Assign a synonym to a path (replace long paths with short names) ■ The path **TNC:\table\WMAT.TAB** is always placed in single quotes ■ The selected synonym is **my_table**
2 Bind QS parameters	Bind a QS parameter to a table column ■ **QS1800** is freely available in NC programs ■ The synonym replaces the entry of the complete path ■ The defined column from the table is called **WMAT**
3 Define search	A search definition contains the entry of the transfer value ■ The **QL1** local parameter (freely selectable) serves to identify the transaction (multiple transactions are possible simultaneously) ■ The synonym defines the table ■ The **WMAT** entry defines the table column of the read operation ■ The entries **NR** 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 control performs the read operation
	■ **SQL FETCH** copies the values from the **result set** into the bound Q or QS parameter
	  ■ **0** successful read operation
	  ■ **1** faulty read operation
	■ The syntax **HANDLE QL1** is the transaction designated by the parameter **QL1**
	■ The parameter **Q1900** is a return value for checking whether the data have been read

5 Complete transaction	The transaction is concluded and the used resources are released

Step		Explanation
6	Remove binding	The binding between table columns and QS parameters is removed (release of necessary resources)
7	Delete synonym	The synonym is deleted (release of necessary resources)

> Synonyms are an alternative only to the required absolute paths. Relative path entries are not possible.

The following NC program shows the entry of an absolute path.

Example: Use an absolute path

0 BEGIN PGM SQL_READ_WMAT_2 MM	
1 SQL BIND QS 1800 "'TNC:\table\WMAT.TAB'.WMAT"	Bind QS parameters
2 SQL QL1 "SELECT WMAT FROM 'TNC:\table\WMAT.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_READ_WMAT_2 MM	

10

Special Functions

10.1 Overview of special functions

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

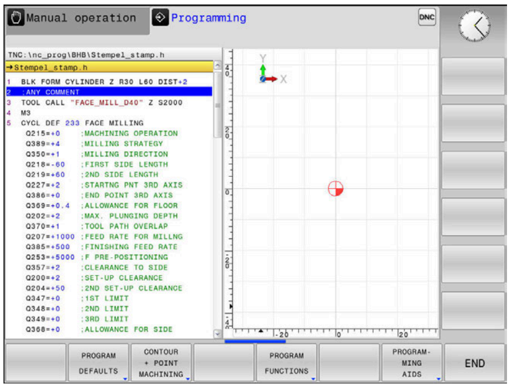
Function	Description
Working with text files	Page 336
Working with freely definable tables	Page 294

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



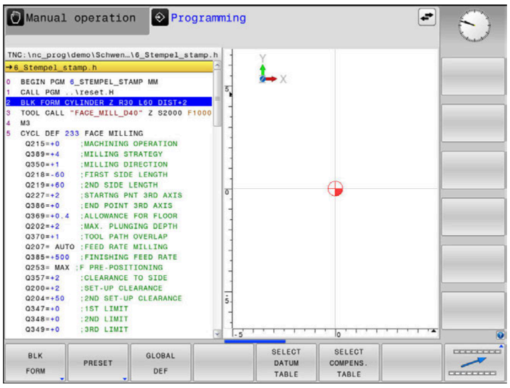
Soft key	Function	Description
FUNCTION MODE	Select machining mode or kinematics	Page 291
PROGRAM DEFAULTS	Define program defaults	Page 289
CONTOUR + POINT MACHINING	Functions for contour and point machining	Page 289
PROGRAM FUNCTIONS	Define different conversational functions	Page 290
PROGRAM-MING AIDS	Programming aids	Page 133

> 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

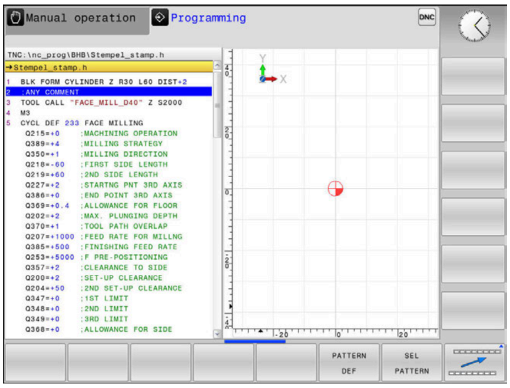


Soft key	Function	Description
BLK FORM	Define workpiece blank	Page 83
PRESET	Modifying the preset	Page 319
SELECT DATUM TABLE	Select datum table	Page 327
SELECT COMPENS. TABLE	Select compensation table	Page 330
GLOBAL DEF	Define global cycle parameters	Page 354

Functions for contour and point machining menu

CONTOUR + POINT MACHINING

▶ Press the soft key for functions for contour and point machining

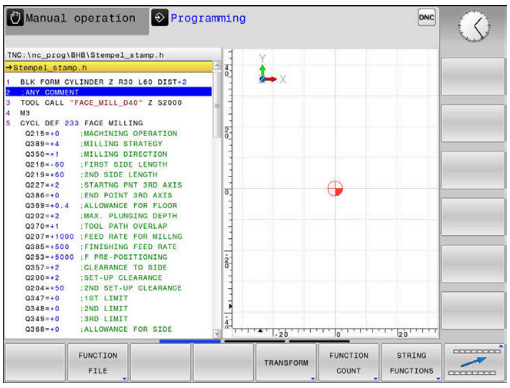


Soft key	Function	Description
PATTERN DEF	Define regular machining pattern	Page 360
SEL PATTERN	Select the point file with machining positions	Page 190

Menu for defining different Klartext functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key



Soft key	Function	Description
FUNCTION FILE	Define file functions	Page 307
TRANSFORM / CORRDATA	Define coordinate transformations Activate compensation values	Page 310 Page 330
FUNCTION COUNT	Define the counter	Page 292
STRING FUNCTIONS	Define string functions	Page 248
FUNCTION SPINDLE	Define pulsing spindle speed	Page 302
FUNCTION FEED	Define recurring dwell time	Page 305
FUNCTION DWELL	Define dwell time in seconds or revolutions	Page 340
INSERT COMMENT	Add comments	Page 137
TABDATA	Write and read table values	Page 332

10.2 Function mode

Program function mode

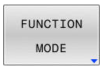
> Refer to your machine manual.
> Your machine manufacturer enables this function.

If your machine manufacturer has enabled the selection of various kinematic models, then you can switch between them using the **FUNCTION MODE** soft key.

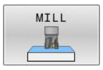
Procedure

To switch the kinematic model:

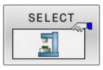
- Show the soft-key row for special functions



- Press the **FUNCTION MODE** soft key



- Press the **MILL** soft key



- Press the **SELECT KINEMATICS** soft key
- Select the desired kinematic model

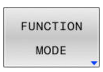
Function Mode Set

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
> Your machine manufacturer defines the available options in the machine parameter **CfgModeSelect** (no. 132200).

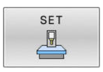
FUNCTION MODE SET allows you to activate settings defined by the machine manufacturer (e.g., changes to the range of traverse) from within the NC program

To select a setting:

- Show the soft-key row with special functions



- Press the **FUNCTION MODE** soft key



- Press the **SET** soft key

- Press the **SELECT** soft key, if required
- > The control opens a selection window.
- Select the desired setting

10.3 Defining a counter

Application

Refer to your machine manual.
Your machine manufacturer enables this function.

With the **FUNCTION COUNT** NC function, you control a counter from within the NC program. This counter allows you, for example, to define a target count up to which the control is to repeat the NC program.

To program this behavior:

SPEC FCT
- Show the soft-key row for 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 counter reading you have defined directly in the NC program is active. The counter reading in the MOD menu remains unaffected.

Effect in the Program Run Single Block and Program Run Full Sequence operating modes

The counter reading from the MOD menu is only active in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.

The counter reading remains the same after a restart of the control.

Defining FUNCTION COUNT

The **FUNCTION COUNT** NC function provides the following counter functions:

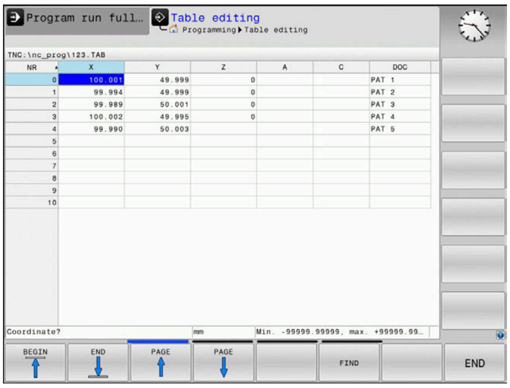
Soft key	Function
FUNCTION COUNT INC	Increase the counter by 1
FUNCTION COUNT RESET	Reset the counter
FUNCTION COUNT TARGET	Define the target count to be reached Input value: 0 to 9999
FUNCTION COUNT SET	Assign a defined value to the counter Input value: 0 to 9999
FUNCTION COUNT ADD	Increase the counter by a defined value Input value: 0 to 9999
FUNCTION COUNT REPEAT	Repeat the NC program from the label if the defined target count has not been reached yet

Example

5 FUNCTION COUNT RESET	Reset the counter reading
6 FUNCTION COUNT TARGET10	Enter the target number of parts to be machined
7 LBL 11	Enter the jump label
8 ...	Machining operation
51 FUNCTION COUNT INC	Increment the counter reading
52 FUNCTION COUNT REPEAT LBL 11	Repeat the machining operations if more parts are to be machined
53 M30	
54 END PGM	

10.4 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 data are input or read.

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 295

> Refer to your machine manual.
> Machine manufacturers 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.

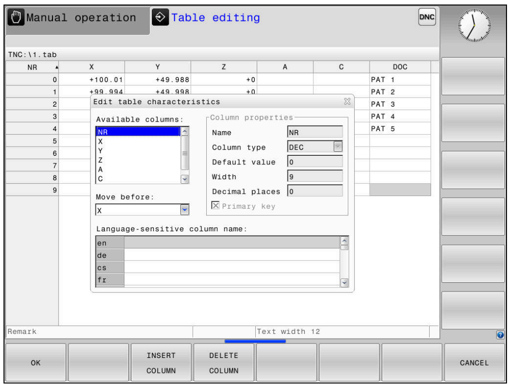
> You can also save your own table templates in the TNC. To do so, create a new table, change the table format and save the table in the **TNC:\system\proto** directory. If you then create a new table, the control offers your template in the selection window for table templates.

Editing the table format

Proceed as follows:

EDIT FORMAT

- 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: Displayed in the header
Column type	**TEXT**: Text entry **SIGN**: Algebraic sign + or - **BIN**: Binary digit **DEC**: Decimal, positive, integer number (cardinal number) **HEX**: Hexadecimal number **INT**: Integer number **LENGTH**: Length (converted in programs with inches) **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**: Permanently defined format for date and time **UPTEXT**: Text entry in uppercase letters **PATHNAME**: Path name
Default value	Default value for the fields in this column
Width	Maximum number of characters in the column The column width is limited as follows: ■ Columns for alphanumeric entries allow up to 100 characters ■ Columns for numeric entries allow up to 15 characters In addition to those 15 characters, the control can display an algebraic sign and a decimal separator
Primary key	First table column
Language-sensitive column name	Language-sensitive dialogs

> 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
- Open the selection menus with the **GOTO** key
- 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.

Closing the structure editor

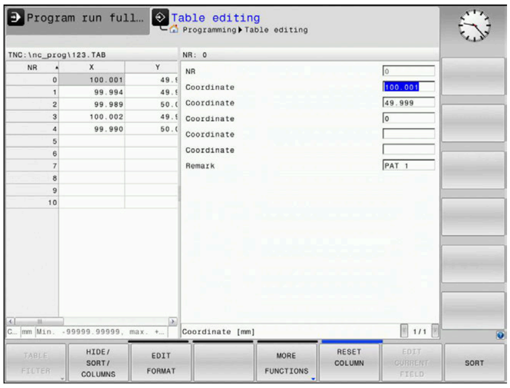
Proceed as follows:

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 Opening a freely definable table

With the **FN 26: TABOPEN** NC function, you open a freely definable table to be written to with **FN 27: TABWRITE** or to be read from with **FN 28: TABREAD**.

> 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**.

11 FN 26: TABOPEN TNC:\table \TAB1.TAB	; Open table with **FN 26**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 26: TABOPEN	Start of syntax for opening a table
File	Path of the table to be opened Fixed or variable name Selection by means of a selection window

Example: Open the table TAB1.TAB, which is saved in the directory TNC:\DIR1.

```
56 FN 26: TABOPEN TNC:\DIR1\TAB1.TAB
```

Use the **SYNTAX** soft key to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path.

Further information: "File names", Page 97

If the complete path is enclosed in quotation marks, you can use both **** and **/** to separate the folders and files.

FN 27: TABWRITE writing to a freely definable table

With the **FN 27: TABWRITE** NC function, you write to the table that you previously opened with **FN 26: TABOPEN**.

Use the **FN 27** NC function to define the table columns to be written to by the control. Within an NC block, you can specify multiple table columns, but only one table row. You can previously define the contents to be written to the columns in variables; or you define it directly in the NC function **FN 27**.

> If you write to multiple columns within one NC block, you need to define the values to be written to the columns in consecutive variables.
>
> If you try to write to a locked or a non-existing table cell, the control displays an error message.
>
> If you write values to multiple columns, the control can either write only numbers or only names.
>
> If you define a fixed value in the **FN 27** NC function, the control will write the same value to each defined column.

Input

11 FN 27: TABWRITE 2/"Length,Radius" = Q2	; Write to table with **FN 27**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 27: TABWRITE	Start of syntax for writing to a table
Number	Row number of the table to be written to Fixed or variable number
Name or **QS**	Column names in the table to be written to Fixed or variable name Use commas to separate multiple column names.
Number, **Name**, or **QS**	Table value Fixed or variable number or name

Example

The control writes to the columns **Radius**, **Depth**, and **D** of row **5** of the currently open table. The control writes the values from the Q parameters **Q5**, **Q6**, and **Q7** to the table.

```
53 Q5 = 3,75
54 Q6 = -5
55 Q7 = 7,5
56 FN 27:  TABWRITE 5/“RADIUS,DEPTH,D“ = Q5
```

FN 28: TABREAD reading a freely definable table

With the **FN 28: TABREAD** NC function, you can read data from the table previously opened with **FN 26: TABOPEN**.

Use the **FN 28** NC function to define the table columns that the control is to read from. Within an NC block, you can specify multiple table columns, but only one table row.

> If you specify multiple columns in an NC block, the control saves the read values in consecutive variables of the same type (e.g., **QL1**, **QL2**, and **QL3**).

Input

11 FN 28: TABREAD Q1 = 2 / "Length"	; Read table with **FN 28**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 28: TABREAD	Start of syntax for reading from a table
Q, **QL**, **QR**, or **QS**	Variable for the source text The control uses this variable to save the contents from the table cells to be read.
Number	Row number in the table to be read Fixed or variable number
Name or **QS**	Column name in the table to be read Fixed or variable name Use commas to separate multiple column names.

Example

The control reads the values of columns **X**, **Y**, and **D** from row **6** of the currently open table. The control saves the values to the Q parameters **Q10**, **Q11**, and **Q12**.

The content from the **DOC** column of the same row is saved to the **QS1** QS parameter.

```
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. The control does not perform an automatic backup of the files prior to a format change. The files will thus be permanently changed and may no longer be usable.

- Only use the function in consultation with the machine manufacturer.

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 data are input or read.

10.5 Pulsing spindle speed FUNCTION S-PULSE

Program pulsing spindle speed

Application

> Refer to your machine manual.
> Read and note the functional description of the machine manufacturer.
> Follow the safety precautions.

Using the **FUNCTION S-PULSE** function, you can program a pulsing spindle speed to avoid natural oscillations of the machine, for example.

With the **P-TIME** input value, you define the duration of an oscillation (oscillation period), and with the **SCALE** input value, the spindle speed change in percent. The spindle speed changes in a sinusoidal form around the nominal value.

Use **FROM-SPEED** and **TO-SPEED** to define the upper and lower spindle speed limits of a spindle speed range in which the pulsing spindle speed is in effect.. Both input values are optional. If you do not define a parameter, the function applies to the entire speed range.

Input

11 FUNCTION S-PULSE P-TIME10 SCALE5 FROM-SPEED4800 TO-SPEED5200	; Spindle speed variation of 5% around the nominal value within 10 seconds (with limit values)

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION S-PULSE	Start of syntax for pulsing spindle speed
P-TIME or **RESET**	Define the duration of an oscillation in seconds, or reset the pulsing spindle speed
SCALE	Spindle speed change in % Only if **P-TIME** has been selected
FROM-SPEED	Lower speed limit from which the pulsing spindle speed will be in effect Only if **P-TIME** has been selected Optional syntax element
TO-SPEED	Upper speed limit up to which the pulsing spindle speed will be in effect Only if **P-TIME** has been selected Optional syntax element

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

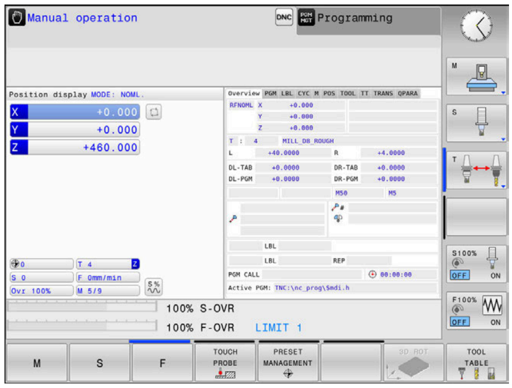
SPINDLE-PULSE
- Press the **SPINDLE-PULSE** soft key
- Define the oscillation period **P-TIME**
- Define the speed change **SCALE**

> The control never exceeds a programmed speed limit. The spindle speed is maintained until the sinusoidal curve of the **FUNCTION S-PULSE** falls below the maximum speed once more.

Icons

In the status bar, the icon indicates the condition of the pulsing spindle speed:

Icon	Function
S % ∿∿	Pulsing spindle speed active



Resetting the pulsing spindle speed

Example

```
18 FUNCTION S-PULSE RESET
```

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

Proceed as follows for the definition:

SPEC FCT	▶ Show the soft-key row with special functions
PROGRAM FUNCTIONS	▶ Press the **PROGRAM FUNCTIONS** soft key
FUNCTION SPINDLE	▶ Press the **FUNCTION SPINDLE** soft key
RESET SPINDLE-PULSE	▶ Press the **RESET SPINDLE-PULSE** soft key.

10.6 Dwell time FUNCTION FEED DWELL

Programming a dwell time

Application

> Refer to your machine manual.
> Read and note the functional description of the machine manufacturer.
> Follow the safety precautions.

FUNCTION FEED DWELL allows you to program a cyclic dwell time in seconds, such as for forcing chip breaking.

Program **FUNCTION FEED DWELL** immediately prior to the operation you wish to run with chip breaking.

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motions.

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, and the spindle continues to turn. During thread cutting, this behavior will cause the workpiece to become scrap. There is also a risk 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 **D-TIME** for dwelling
- Define the interval duration **F-TIME** for cutting

Resetting the dwell time

> Reset 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.

10.7 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.

Programming and operating information:

- 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**.
- The **FUNCTION FILE** function is considered only in the **Program run, single block** and **Program run, full sequence** operating modes.

Defining file functions

Proceed as follows:

SPEC FCT

- ▶ Press the special functions key

PROGRAM FUNCTIONS

- ▶ Select the program functions

FUNCTION FILE

- ▶ Select file operations
- > The control displays the available functions.

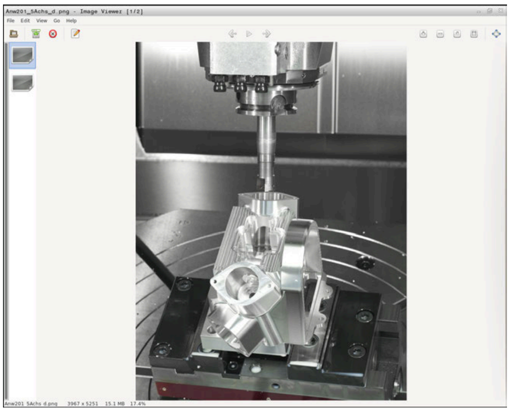
Soft key	Function	Meaning
FILE COPY	**FILE COPY**	Copy file: Enter the name and path of the file to be copied, as well as the target path
FILE MOVE	**FILE MOVE**	Move file: Enter the name and path of the file to be moved, as well as the target path
FILE DELETE	**FILE DELETE**	Delete file: Enter the path and name of the file to be deleted
OPEN FILE	**OPEN FILE**	Open the file: Enter the name and path of the file

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.

OPEN FILE

Fundamentals



The **OPEN FILE** function allows you to open various file types directly from within the NC program.

If you define **OPEN FILE**, the control continues the dialog and you can program a **STOP**.

Using this function, the control can open all file types that you can open manually.

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

The control opens the file in the software tool last used for this file type. If you have never opened a file of a certain file type and multiple software tools are available, the control will interrupt program run and open the **Application?** window. In the **Application?** window, you can select the software tool the control should use to open the file. The control saves this selection.

Multiple software tools are available for opening the following file types:

- CFG
- SVG
- BMP
- GIF
- JPG/JPEG
- PNG

> In order to avoid program run interruptions or having to select an alternative software tool, open a file of the corresponding file type once in the file manager. If the files of a certain file type can be opened in multiple software tools, you can use the file manager to select the software tool to be used for opening files of this file type.
>
> **Further information:** User's Manual for **Setup, Testing and Running NC Programs**

The **OPEN FILE** function is available in the following operating modes:

- **Positioning w/ Manual Data Input**
- **Test Run**
- **Program Run Single Block**
- **Program Run Full Sequence**

Programming OPEN FILE

To program the **OPEN FILE** function:

SPEC FCT
- ▶ Press the special functions key

PROGRAM FUNCTIONS
- ▶ Select the program functions

FUNCTION FILE
- ▶ Select file operations

OPEN FILE
- ▶ Select the **OPEN FILE** function
- > The control initiates the dialog.

SELECT FILE
- ▶ Press the **SELECT FILE** soft key
- ▶ In the folder structure, select the file to be displayed

OK
- ▶ Press the **OK** soft key.
- > The control displays the path of the selected file and the **STOP** function.
- ▶ Optionally, program **STOP**
- > The control concludes the entry of the **OPEN FILE** function.

Automatic display

For the display of some file types, the control provides only one additional tool. With the **OPEN FILE** function, the control then automatically uses this tool to display files of these formats.

Example

```
1 OPEN FILE "TNC:\CLAMPING_INFORMATION.HTML"
```

HEROS tool that can be used for displaying:

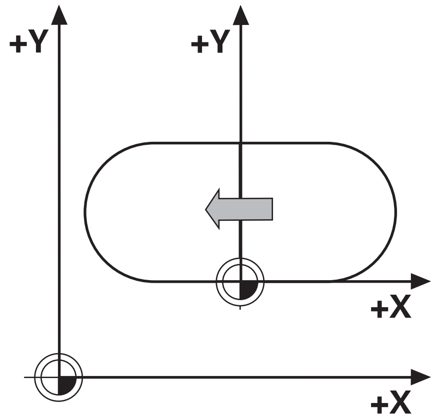
- Mozilla Firefox

10.8 NC functions for coordinate transformations

Overview

The control provides the following **TRANS** functions:

Syntax	Meaning	Further information
TRANS DATUM	Shift the workpiece datum	Page 310
TRANS MIRROR	Mirror an axis	Page 313
TRANS SCALE	Scale contours and positions	Page 315
TRANS RESET	Reset the coordinate transformation	Page 316



Define the functions in the sequence in which they are listed in the table and reset them in reverse order. The sequence of programming will have an impact on the result.

For example, if you first shift the workpiece datum and then mirror the contour and then reverse the sequence, the contour will be mirrored at the original workpiece datum.

All **TRANS** functions reference the workpiece datum. The workpiece datum is the origin of the input coordinate system (**I-CS**).

Further information: "Input coordinate system I-CS", Page

Related topics

- Coordinate transformation cycles

  Further information: User's Manual for **Programming of Machining Cycles**
- Reference systems

  Further information: "Reference system of milling machines", Page 79

Datum shift with TRANS DATUM

Application

The **TRANS DATUM** function allows you to shift the workpiece datum by either entering fixed or variable coordinates or by specifying a table row in the datum table.

Use the **TRANS DATUM RESET** function to reset the datum shift.

Related topics

- Activating the datum table

  Further information: User's Manual for **Programming of Machining Cycles**

Description of function

TRANS DATUM AXIS
You can define a datum shift by entering values in the respective axis with the **TRANS DATUM AXIS** function. You can define up to nine coordinates in one NC block, and incremental entries are possible.

If a datum shift is active, the control displays it on the **TRANS** tab of the additional status display.

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

The control displays the result of the datum shift in the position display.

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

TRANS DATUM TABLE
You can use the **TRANS DATUM TABLE** function to define a datum shift by selecting a row from a datum table.

Optionally, you can set the path to a datum table. If you do not define a path, the control will use the datum table that has been activated with **SEL TABLE**.

Further information: "Activating the datum table in your NC program", Page 327

The control displays the datum shift with **TRANS DATUM TABLE** and the path to the datum table on the **TRANS** tab of the additional status display.

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

TRANS DATUM RESET
Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant.

Input

11 TRANS DATUM AXIS X+10 Y +25 Z+42	; Shift the workpiece datum in the **X**, **Y** and **Z** axes

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Functions** ▶ **TRANSFORM** ▶ **TRANS DATUM**

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS DATUM	Start of syntax for a datum shift
AXIS, **TABLE** or **RESET**	Datum shift with coordinate input, with a datum table or reset of the datum shift
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V** or **W**	Possible axes for coordinate input Fixed or variable number Only if **AXIS** has been selected
TABLINE	Row in the datum table Fixed or variable number Only if **TABLE** has been selected
Name or **QS**	Path to the datum table Fixed or variable path Selection by means of a selection window Optional syntax element Only if **TABLE** has been selected

Notes

- Absolute values reference the workpiece preset. Incremental values reference the workpiece datum.
- If you execute an absolute datum shift with **TRANS DATUM** or Cycle **7 DATUM SHIFT**, then the control overwrites the values of the current datum shift. The control adds the incremental values to the values of the current datum shift.

  Further information: User's Manual for **Programming of Machining Cycles**
- A datum shift in the axes **A**, **B**, **C**, **U**, **V** and **W** is effective as an offset. HEIDENHAIN recommends inclining rotary axes using the **PLANE** functions or a 3D basic rotation.

  Further information: User's Manual for **Setup, Testing and Running NC Programs**
- In machine parameter **transDatumCoordSys** (no. 127501), the machine manufacturer defines the reference system referred to by the values in the position display.
- 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**).

Mirroring with TRANS MIRROR

Application

Use the **TRANS MIRROR** function to mirror contours or positions about one or more axes.

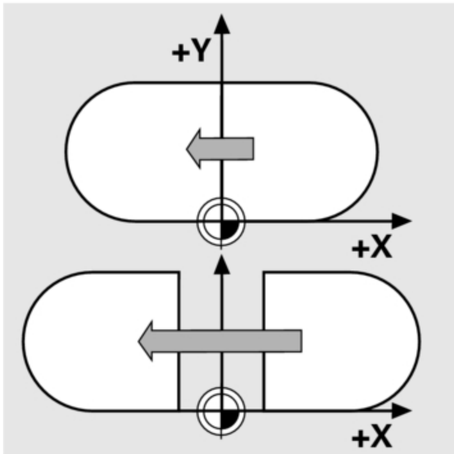
The **TRANS MIRROR RESET** function allows you to reset mirroring.

Related topics

- Cycle **8 MIRRORING**

  Further information: User's Manual for **Programming of Machining Cycles**

Description of function



Mirroring is a modal function that in effect as soon as it has been defined in the NC program.

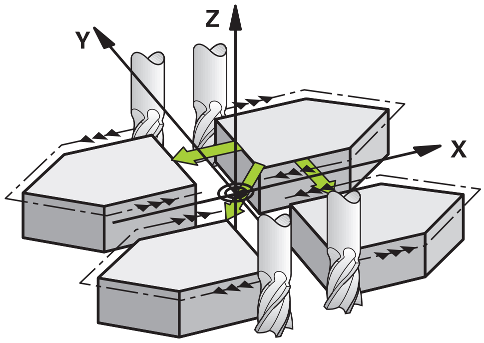
The control mirrors contours or positions about the active workpiece datum. If the datum is outside the contour, the control will also mirror the distance to the datum.

If you mirror only one axis, the machining direction of the tool is reversed. The rotational direction defined in a cycle will remain unchanged (e.g., if defined within one of the OCM cycles).

Depending on the selected **AXIS** axis values, the control will mirror the following working planes:

- **X**: The control mirrors the **YZ** working plane
- **Y**: The control mirrors the **ZX** working plane
- **Z**: The control mirrors the **XY** working plane

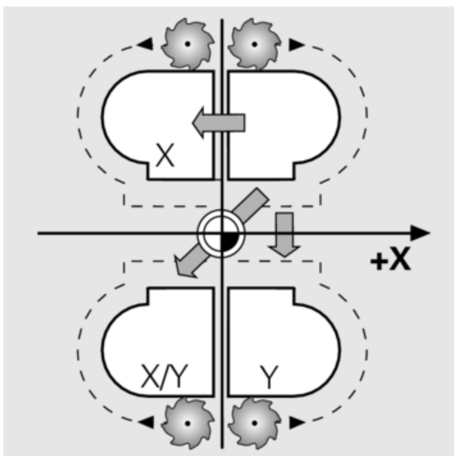
Further information: "Designation of the axes on milling machines", Page 79



You can select up to three axis values.

If mirroring is active, the control displays it on the **TRANS** tab of the additional status display.

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



Input

11 TRANS MIRROR AXIS X	; Mirror X coordinates about the Y axis

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS MIRROR	Start of syntax for mirroring
AXIS or **RESET**	Enter mirroring of axis values or reset mirroring
X, **Y** or **Z**	Axis values to be mirrored Only if **AXIS** has been selected

Notes

- This function can only be used in the **FUNCTION MODE MILL** machining mode.

  Further information: User's Manual for **Setup, Testing and Running NC Programs**
- If you execute mirroring with **TRANS MIRROR** or Cycle **8 MIRRORING**, then the control overwrites the current mirroring.

  Further information: User's Manual for **Programming of Machining Cycles**

Notes on using these functions in conjunction with tilting functions

NOTICE

Danger of collision!

The control reacts differently to the various types of transformations as well as their programmed sequence. Unexpected movements or collisions can occur if the functions are not suitable.

- Program only the recommended transformations in the respective reference system
- Use tilting functions with spatial angles instead of with axis angles
- Use the Simulation mode to test the NC program

The type of tilting function has the following effects on the result:

- If you tilt using spatial angles (**PLANE** functions except for **PLANE AXIAL** or Cycle **19**), previously programmed transformations will change the position of the workpiece datum and the orientation of the rotary axes:
  - Shifting with the **TRANS DATUM** function will change the position of the workpiece datum.
  - Mirroring changes the orientation of the rotary axes. The entire NC program, including the spatial angles, will be mirrored.
- If you tilt using axis angles (**PLANE AXIAL** or Cycle **19**), a previously programmed mirroring has no effect on the orientation of the rotary axes. You use these functions for direct positioning of the machine axes.

Further information: "Workpiece coordinate system W-CS", Page

Scaling with TRANS SCALE

Application

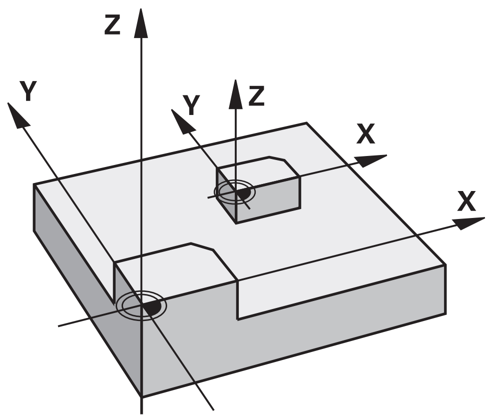
The **TRANS SCALE** function lets you change the scale of the contours or distances to the datum, thereby evenly enlarging or shrinking them. This enables you to program shrinkage and oversize allowances, for example.

Use the **TRANS SCALE RESET** function to reset scaling.

Related topics

- Cycle **11 SCALING FACTOR**

  Further information: User's Manual for **Programming of Machining Cycles**



Description of function

Scaling is a modal function that is in effect as soon as it has been defined in the NC program.

Depending on the position of the workpiece datum, scaling is carried out as follows:

- Workpiece datum at the center of the contour:
  The contour is scaled uniformly in all directions.
- Workpiece datum at the bottom left of the contour:
  The contour is scaled in the positive X and Y axis directions.
- Workpiece datum at the top right of the contour:
  The contour is scaled in the negative X and Y axis directions.

If you enter a scaling factor **SCL** less than 1, the contour will be reduced in size. If you enter a scaling factor **SCL** greater than 1, the contour will be enlarged.

When scaling, the control takes the coordinate input and dimensions from all cycles into account.

If scaling is active, the control displays it on the **TRANS** tab of the additional status display.

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

Input

11 TRANS SCALE SCL1.5	; Enlarge the contour by the factor 1.5

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS SCALE	Start of syntax for scaling
SCL or **RESET**	Enter the scaling factor or reset scaling Fixed or variable number

Notes

- This function can only be used in the **FUNCTION MODE MILL** machining mode.
  Further information: User's Manual for **Setup, Testing and Running NC Programs**
- If you execute a change of scale with **TRANS SCALE** or Cycle **11 SCALING FACTOR**, then the control overwrites the current scaling factor.
  Further information: User's Manual for **Programming of Machining Cycles**
- If you want to reduce the size of a contour with inside radii, make sure to select an appropriate tool. Otherwise, residual material might remain.

Resetting with TRANS RESET

Application

Use the NC function **TRANS RESET** to reset all simple coordinate transformations simultaneously.

Related topics

- NC functions for coordinate transformation

  Further information: "NC functions for coordinate transformations", Page 310
- Coordinate transformation cycles

  Further information: User's Manual for **Programming of Machining Cycles**

Description of function

The control resets all of the following simple coordinate transformations:

Coordinate transformation	Syntax	Further information
Datum shift	**TRANS DATUM**	Page 310
	Cycle **7 DATUM SHIFT**	See the User's Manual for Programming of Machining Cycles
Mirroring	**TRANS MIRROR**	Page 313
	Cycle **8 MIRRORING**	See the User's Manual for Programming of Machining Cycles
Scaling	**TRANS SCALE**	Page 315
	Cycle **11 SCALING FACTOR**	See the User's Manual for Programming of Machining Cycles
	Cycle **26 AXIS-SPECIFIC SCALING**	See the User's Manual for Programming of Machining Cycles

> The control also resets simple coordinate transformations defined by the machine manufacturer.

Input

```
11 TRANS RESET    ; Reset simple coordinate transformations
```

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS RESET	Syntax opener for resetting simple coordinate transformations

Selecting a TRANS function

To select a **TRANS** function:

SPEC FCT
- Show the soft-key row with special functions

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

TRANSFORM / CORRDATA
- Press the **TRANSFORM / CORRDATA** soft key

TRANSFOR-MATIONS
- Press the **TRANSFORMATIONS** soft key

- Press the soft key for the desired **TRANS** function

10.9 Modifying presets

The control provides the following functions for modifying a preset directly in the NC program after it has been defined in the preset table:

- Activate the preset
- Copy the preset
- Correct the preset

Activating a preset

The **PRESET SELECT** function allows you to use a preset defined in the preset table and activate it as a new preset.

To activate the preset, use the row number or the content in the **DOC** column.

NOTICE

Danger of collision!

Depending on the machine parameter **CfgColumnDescription** (no. 105607), you can define the same content several times in the **DOC** column of the preset table. In this case, if you activate a preset using the **DOC** column, the control selects the preset with the lowest row number. If the control does not select the desired preset there is a risk of collision.

- ▶ Uniquely define the content of the **DOC** column
- ▶ Only activate the preset with the row number

If you program **PRESET SELECT** without optional parameters, then the behavior is identical to Cycle **247 PRESETTING**.

Use the optional parameters to define the following:

- **KEEP TRANS**: Retain simple transformations
  - Cycle **7 DATUM SHIFT**
  - Cycle **8 MIRRORING**
  - Cycle **11 SCALING FACTOR**
  - Cycle **26 AXIS-SPECIFIC SCALING**
- **WP**: Any changes apply to the workpiece preset

Procedure

Proceed as follows for the definition:

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM DEFAULTS

- Press the **PROGRAM DEFAULTS** soft key

PRESET

- Press the **PRESET** soft key

PRESET SELECT

- Press the **PRESET SELECT** soft key
- Define the desired preset number
- Alternatively, define the entry from the **DOC** column
- Retain the transformations where necessary
- If necessary, select the preset to which the change is to apply

Example

13 PRESET SELECT #3 KEEP TRANS WP	Select Preset 3 as the workpiece preset, and retain the transformations

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- Before activating a preset, check whether all columns contain values.
- For undefined columns, enter values (e.g., **0**)
- As an alternative, have the machine manufacturer define **0** as the default value for the columns

Copying a preset

The function **PRESET COPY** allows you to copy a preset defined in the preset table and activate the preset copied.

To select the preset to be copied, use the row number or the entry in the **DOC** column.

Use the optional parameters to define the following:

- **SELECT TARGET**: Activate the copied preset
- **KEEP TRANS**: Retain simple transformations

NOTICE

Danger of collision!

Depending on the machine parameter **CfgColumnDescription** (no. 105607), you can define the same content several times in the **DOC** column of the preset table. In this case, if you activate a preset using the **DOC** column, the control selects the preset with the lowest row number. If the control does not select the desired preset there is a risk of collision.

- Uniquely define the content of the **DOC** column
- Only activate the preset with the row number

Procedure

Proceed as follows for the definition:

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

PROGRAM DEFAULTS
- Press the **PROGRAM DEFAULTS** soft key

PRESET
- Press the **PRESET** soft key

PRESET COPY
- Press the **PRESET COPY** soft key
- Define the preset number to be copied
- Alternatively, define the entry from the **DOC** column
- Define the new preset number
- Activate the copied preset, if necessary
- Retain the transformations where necessary

Example

13 PRESET COPY #1 TO #3 SELECT TARGET KEEP TRANS	Copy the preset 1 to line 3, activate the preset 3, and retain the transformations

Correcting a preset

The function **PRESET CORR** allows you to correct the active preset.

If both the basic rotation and a translation are corrected in an NC block, the control will first correct the translation and then the basic rotation.

The compensation values are given with respect to the active coordinate system.

Procedure

Proceed as follows for the definition:

SPEC FCT
- Show the soft key row with special functions

PROGRAM DEFAULTS
- Press the **PROGRAM DEFAULTS** soft key

PRESET
- Press the **PRESET** soft key

PRESET CORR
- Press the **PRESET CORR** soft key

- Define the desired compensation values

Example

13 PRESET CORR X+10 SPC+45	The active preset is corrected by a value of +10 mm in X, and by +45° in SPC

10.10 Datum table

Application

You can save the workpiece-related datums in a datum table. To use a datum table, you must activate it.

Description

Datums from a datum table always reference the current preset. The coordinate values from datum tables are only effective as absolute coordinate values.

Use datum tables for the following purposes:

- Frequent use of the same datum shift
- Frequently recurring machining sequences on the workpiece
- Frequently recurring machining sequences at various locations on the workpiece

The values of the columns **X**, **Y** and **Z** are applied as shifts in the workpiece coordinate system **W-CS**. The values of the columns **A**, **B**, **C**, **U**, **V** and **W** are applied as shifts in the machine coordinate system **M-CS**.

The datum table contains the following parameters:

Parameter	Meaning	Input
D	Sequential number of the datums	**0...99999999**
X	X coordinate of the datum	**-99999.99999...99999.99999**
Y	Y coordinate of the datum	**-99999.99999...99999.99999**
Z	Z coordinate of the datum	**-99999.99999...99999.99999**
A	Axis angle of the A axis for the datum	**-360.0000000...360.0000000**
B	Axis angle of the B axis for the datum	**-360.0000000...360.0000000**
C	Axis angle of the C axis for the datum	**-360.0000000...360.0000000**
U	Position of the U axis for the datum	**-99999.99999...99999.99999**
V	Position of the V axis for the datum	**-99999.99999...99999.99999**
W	Position of the W axis for the datum	**-99999.99999...99999.99999**
DOC	Comment column	Max. 16 characters

Creating a datum table

To create a new datum table:

- Switch to the **Programming** operating mode
- Press the **PGM MGT** key
- Press the **NEW FILE** soft key
- > The control opens the **New file** window where you can enter the file name.
- Enter the file name with the file type ***.d**
- Confirm with the **ENT** key
- > The control opens the **Select table format** window, if necessary.
- Select a table format, if necessary
- Press the **OK** soft key, if necessary
- Select unit of measure **MM** or **INCH**, if necessary
- > The control opens the datum table.

> You can select the table format, if there is at least one prototype of the table type.
> The control displays whether the prototype is defined to use mm or inches as unit of measure. If the control displays both units of measure, you can select one of them.
> The machine manufacturer defines the prototypes.

> 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 data are input or read.
> **Further information:** "Accessing tables with SQL statements", Page 264

Opening and editing a datum table

> **i** 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.

To open and edit a datum table:

PGM MGT
- ▶ Press the **PGM MGT** key
- ▶ Select the desired datum table
- > The control opens the datum table.
- ▶ Select the row you wish to edit

ENT
- ▶ Save your input, e.g. by pressing the **ENT** key.

> **i** To delete the value from the input field, press the **CE** key.

The control displays the following functions in the soft-key row:

Soft key	Function
BEGIN	Select the table start
END	Select the table end
PAGE	Go to previous page
PAGE	Go to next page
FIND	Search The control opens a window where you can enter the text or value you are looking for.
RESET TABLE	Reset table
BEGIN LINE	Move the cursor to the beginning of the row
END LINE	Move the cursor to the end of the row
COPY FIELD	Copy the current value
PASTE FIELD	Paste the copied value
APPEND N LINES AT END	Insert the specified number of rows New rows can only be inserted at the end of the table.

Soft key	Function
INSERT LINE	Insert row New rows can only be inserted at the end of the table.
DELETE LINE	Delete row
SORT/ HIDE COLUMNS	Sort/hide columns The control opens the **Column sequence** window with the following options: ■ **Use standard format** ■ Display/hide columns ■ Arrange columns ■ Freeze columns (3 max.)
MORE FUNCTIONS	Additional functions, e.g. Delete
RESET COLUMN	Reset the column
EDIT CURRENT FIELD	Edit the current field
SORT	Sort the datum table A window opens where you can select the sorting order.

> If you enter the code number 555343, the control will display the **EDIT FORMAT** soft key. With this soft key, you can change the table properties.

Activating the datum table in your NC program

To activate a workpiece datum table in your NC program:

PGM CALL
- Press the **PGM CALL** key

SELECT DATUM TABLE
- Press the **SELECT DATUM TABLE** soft key

SELECT FILE
- Press the **SELECT FILE** soft key
- > A file selection window opens.
- Select the desired datum table

ENT
- Confirm with the **ENT** key

> If you enter the datum table name manually, please note the following:
> - If the datum table is located in the same directory as the NC program, enter the file name only.
> - If the datum table is not located in the same directory as the NC program, enter the complete path.

> Program **SEL TABLE** before Cycle **7** or the **TRANS DATUM** function.

Activating the datum table manually

> If you do not use **SEL TABLE**, you must activate the desired datum table prior to the test run.

To activate a datum table for the test run:

- Select the **Test Run** operating mode

PGM MGT
- Press the **PGM MGT** key
- Select the desired datum table
- > The control activates the datum table for the test run and marks the file with the **S** status.

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

10.11 Compensation table

Application

With the compensation table, you can save compensations in the tool coordinate system (T-CS) or in the working plane coordinate system (WPL-CS).

The compensation table **.tco** is the alternative to compensating with **DL**, **DR** and **DR2** in the Tool Call block. As soon as you have activated a compensation table, the control overwrites the compensation value from the Tool Call block.

The compensation tables offer the following benefits:

- Values can be changed without adapting the NC program
- Values can be changed during NC program run

If you change a value, then this change does not become active until the compensation is called again.

Types of compensation tables

Via the file name extension, you can determine in which coordinate system the control will perform the compensation.

The control provides the following compensation tables:

- tco (tool correction): Compensation in the tool coordinate system (**T-CS**)
- wco (workpiece correction): Compensation in the working plane coordinate system (**WPL-CS**)

Compensation via the table is an alternative to the compensation in the **TOOL CALL** block. Compensation from the table overwrites an already programmed compensation in the **TOOL CALL** block.

Compensation in the tool coordinate system (T-CS)

Any compensation in the compensation tables with the ***.tco** file name extension applies to the active tool. The table applies to all tool types. Therefore, columns that you may not need for your specific tool type will be displayed during creation.

> Enter only those values that are relevant to your tool. If you compensate for values that are not present with the existing tool, the control issues an error message.

The compensations have the following effects:

- In the case of milling cutters, as an alternative to the delta values in the **TOOL CALL**

If a shift with the ***.tco** compensation table is active, the control displays it on the **TOOL** tab of the additional status display.

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

Compensation in the working plane coordinate system (WPL-CS)

The values from the compensation tables with the ***.wco** file name extension are applied as shifts in the working plane coordinate system (**WPL-CS**).

Creating a compensation table

Before you can work with a compensation table, you must first create the respective table.

You can create a compensation table as follows:

- Switch to the **Programming** operating mode
- Press the **PGM MGT** key
- Press the **NEW FILE** soft key
- Enter a file name with the desired extension (e.g., Corr.tco)
- Confirm by pressing the **ENT** key
- > The control opens the **Select table format** window, if necessary.
- Select a table format, if necessary
- Press the **OK** soft key, if necessary
- Select **MM** or **INCH** as the unit of measure, if necessary
- > The control opens the compensation table.
- Press the **APPEND N LINES AT END** soft key
- Enter the compensation values

You can select the table format, if there is at least one prototype of the table type.

The control displays whether the prototype is defined to use mm or inches as unit of measure. If the control displays both units of measure, you can select one of them.

The machine manufacturer defines the prototypes.

Activate the compensation table

Selecting a compensation table

If you are using compensation tables, then use the **SEL CORR-TABLE** function to activate the desired compensation table from within the NC program.

To add a compensation table to the NC program:

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM DEFAULTS

- Press the **PROGRAM DEFAULTS** soft key

SELECT COMPENS. TABLE

- Press the **SELECT COMPENS. TABLE** soft key

TCS

- Press the soft key of the table type (e.g., **TCS**
- Select the table

If you are working without the **SEL CORR-TABLE** function, then you must activate the desired table prior to the test run or program run.

In all operating modes, proceed as follows:

- Select the desired operating mode
- Select the desired table in the file manager
- > In the **Test Run** operating mode, the table receives the status S; in the **Program run, single block** and **Program run, full sequence** operating modes, it receives the status M.

Activating a compensation value

To activate a compensation value in the NC program:

SPEC FCT

- Press the **SPEC FCT** key

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

TRANSFORM / CORRDATA

- Press the **TRANSFORM / CORRDATA** soft key

FUNCTION CORRDATA

- Press the **FUNCTION CORRDATA** soft key

TCS

- Press the soft key of the desired compensation (e.g., **TCS**
- Enter the line number

Duration of active compensation

Activated compensation stays in effect until the end of the program or until a tool change occurs.

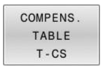
With **FUNCTION CORRDATA RESET**, you can program the compensations to reset.

Editing a compensation table during program run

You can change the values in the active compensation table during program run. As long as the compensation table is not yet active, the control dims the soft key.

Proceed as follows:

- Press the **OPEN COMPENS. TABLES** soft key



- Press the soft key of the desired table (e.g., **COMPENS. TABLE T-CS**)

- Set the **EDIT** soft key to **ON**
- Use the arrow keys to navigate to the desired location
- Edit the value

> The changed data do not take effect until after the compensation has been activated again.

10.12 Accessing table values

Application

The **TABDATA** functions allow you to access table values.

These functions enable automated editing of compensation values from within the NC program, for example.

You can access the following tables:

- Tool table ***.t** (read-only access)
- Compensation table ***.tco** (read and write access)
- Compensation table ***.wco** (read and write access)
- Preset table ***.pr** (read and write access)

In each case, the active table is accessed. Read-only access is always possible, whereas write access is possible only during program run. Write access during simulation or during a block scan has no effect.

If the unit of measure used in the NC program differs from that used in the table, the control converts the values from **millimeters** to **inches**, and vice versa.

Reading a table value

The function **TABDATA READ** allows you to read a value from a table and save it to a Q parameter.

Depending on the type of column you want to transfer, you can use **Q**, **QL**, **QR**, or **QS** to save the value. The control automatically converts the table values to the unit of measure used in the NC program.

The control reads from the currently active tool table and preset table. You can read a value from a compensation table only if you have activated the table concerned.

For example, the **TABDATA READ** function enables you to pre-check the data of the tool to be used to prevent error messages from occurring during program run.

Procedure

Proceed as follows:

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

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

TABDATA
- Press the **TABDATA** soft key

TABDATA READ
- Press the **TABDATA READ** soft key
- Enter the Q parameter for the result

ENT
- Confirm with the **ENT** key

CORR-TCS
- Press the soft key for the desired table (e.g., **CORR-TCS**)
- Enter the column name

ENT
- Confirm with the **ENT** key
- Enter the row number of the table

ENT
- Press the **ENT** key

Example

12 SEL CORR-TABLE TCS "TNC:\table\corr.tco"	Activate the compensation table
13 TABDATA READ Q1 = CORR-TCS COLUMN "DR" KEY "5"	Save the value of row 5, column DR, from the compensation table to Q1

Writing a table value

Use the function **TABDATA WRITE** to write a value into a table.

Depending on the type of column you want to write to, you can use **Q**, **QL**, **QR**, or **QS** as a transfer parameter. Alternatively, you can define the value directly in the NC function **TABDATA WRITE**.

In order to write into a compensation table, you need to activate the table.

You can use the **TABDATA WRITE** function after a touch probe cycle to enter a necessary tool compensation into the compensation table, for example.

Procedure

Proceed as follows:

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

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

TABDATA
- Press the **TABDATA** soft key

TABDATA WRITE
- Press the **TABDATA WRITE** soft key

CORR-TCS
- Press the soft key for the desired table (e.g., **CORR-TCS**)
- Enter the column name

ENT
- Confirm with the **ENT** key
- Enter the row number of the table

ENT
- Confirm with the **ENT** key
- Enter the number, name or variable

ENT
- Confirm with the **ENT** key

Example

12 SEL CORR-TABLE TCS "TNC:\table\corr.tco"	Activate the compensation table
13 TABDATA WRITE CORR-TCS COLUMN "DR" KEY "3" = Q1	Write the value from Q1 into line 3, column DR, of the compensation table

Adding a table value

Use the **TABDATA ADD** function to add a value to an existing table value.

Depending on the type of column you want to write to, you can use **Q**, **QL**, or **QR** as a transfer parameter. Alternatively, you can define the value directly in the NC function **TABDATA ADD**.

In order to write into a compensation table, you need to activate the table.

You can use the **TABDATA ADD** function to update a tool compensation value after a measurement has been repeated, for example.

Procedure

Proceed as follows:

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

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

TABDATA
- Press the **TABDATA** soft key

TABDATA ADDITION
- Press the **TABDATA ADDITION** soft key

CORR-TCS
- Press the soft key for the desired table (e.g., **CORR-TCS**)
- Enter the column name

ENT
- Confirm with the **ENT** key
- Enter the row number of the table

ENT
- Confirm with the **ENT** key
- Enter the number or variable

ENT
- Confirm with the **ENT** key

Example

12 SEL CORR-TABLE TCS "TNC:\table\corr.tco"	Activate the compensation table
13 TABDATA ADD CORR-TCS COLUMN "DR" KEY "3" = Q1	Add the value from Q1 to line 3, column DR, of the compensation table

10.13 Creating text files

Application

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

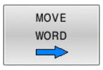
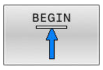
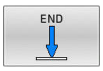
- Recording test results
- Documenting working procedures
- Creating formula collections

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 then the **SHOW ALL** soft key
- Select a file and open it with the **SELECT** soft key or **ENT** key, or open 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 removed and buffered
- Move the cursor to the location where you wish to 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 prompt.
- 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 prompt.
- 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.
- Select the search function: Press the **FIND** soft key
- Press the **FIND CURRENT WORD** soft key
- Find a word: Press the **FIND** soft key
- Exit the search function: Press the **END** soft key

Finding any text

- To select the search function, press the **FIND** soft key. The control shows the **Find text :** dialog prompt
- Enter the text that you wish to find
- Find text: Press the **FIND** soft key
- Exit the search function: Press the **END** soft key

10.14 Dwell time FUNCTION DWELL

Programming a dwell time

Application

The **FUNCTION DWELL** function allows 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

- Press the **FUNCTION DWELL** soft key

DWELL TIME

- Press the **DWELL TIME** soft key
- Define the duration in seconds

DWELL REVOLUTIONS

- Or press the **DWELL REVOLUTIONS** soft key
- Define the number of spindle revolutions

11

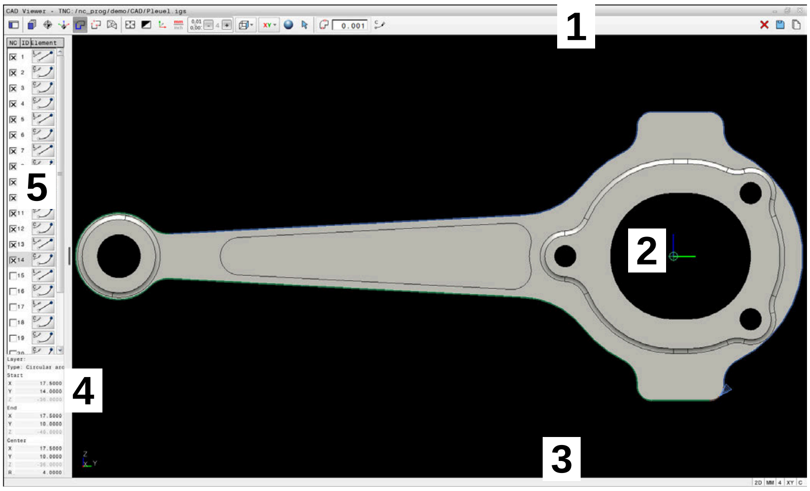
CAD Viewer

11.1 Screen layout of CAD Viewer

CAD Viewer fundamentals

Screen display

When you open **CAD Viewer**, the following screen layout is displayed:



1 Menu bar
2 Graphics area
3 Status bar
4 Element information area
5 List View area

File types

CAD Viewer supports the following standard file types that can be opened directly in the control:

File type	Extension	Format
STEP	*.stp and *.step	■ AP 203 ■ AP 214
IGES	*.igs and *.iges	■ Version 5.3
DXF	*.dxf	■ R10 to 2015 ■ ASCII
STL	*.stl	■ Binary ■ ASCII

CAD Viewer allows you to open CAD files consisting of any number of triangles.

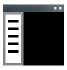
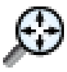
11.2 CAD Viewer

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

The control’s full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

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!

- Test your program before executing it

If you use indirect parameter assignments in cycles with numbers greater than **200** (e.g., **Q210 = Q1**), any change in the assigned paramete (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 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 machining cycles

CYCL DEF

- Press the **CYCL DEF** key

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring, tapping and counterboring	379
POCKETS/ STUDS/ SLOTS	Cycles for milling rectangular pockets and studs, slots, and face milling	437
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	469
PATTERN	Cycles for producing point patterns	370
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop,	481

▷

- If required, switch to machine-specific machining cycles

  The machine manufacturer can integrate these types of machining cycles.

12.3 Working with fixed cycles

Machine-specific cycles

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

Cycles are available for many machines. Your machine manufacturer can implement 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

NOTICE

Danger of collision!

HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. Using variables outside the recommended ranges can lead to intersections and thus, undesired behavior. Danger of collision during machining!

- Only use variable ranges recommended by HEIDENHAIN
- Do not use pre-assigned variables
- Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers
- Check the machining sequence using the simulation

Further information: "Calling a cycle", Page 351

Defining a cycle using soft keys

Proceed as follows:

CYCL DEF

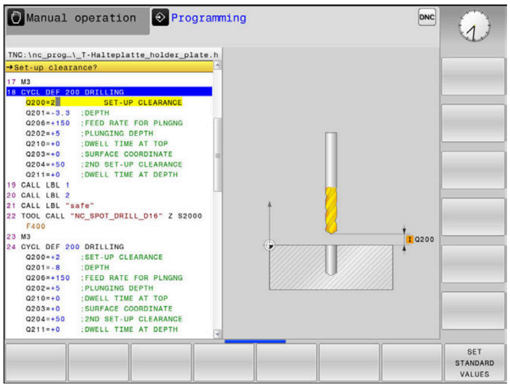
- ▶ Press the **CYCL DEF** key
- > The soft-key row shows the available groups of cycles.

DRILLING/ THREAD

- ▶ Select the desired cycle group (e.g., drilling cycles)

262

- ▶ Select the desired cycle (e.g., Cycle **200 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 the required parameters
- ▶ Conclude each input with the **ENT** key
- > The control closes the dialog when all required data has been entered.



NOTICE

Danger of collision!

You can program variables as input values in HEIDENHAIN cycles. Using variables outside of the recommended input ranges can lead to collisions.

- ▶ Only use the input ranges recommended by HEIDENHAIN
- ▶ Pay attention to the HEIDENHAIN documentation
- ▶ Check the machining sequence using a simulation

Defining a cycle using the GOTO function

Proceed as follows:

CYCL DEF

- ▶ Press the **CYCL DEF** key
- > The soft-key row shows the available groups of cycles.

GOTO □

- ▶ Press the **GOTO** key
- > 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

```
11  CYCL DEF 200 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
```

Calling a cycle

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (only required for test graphics)
- Tool call
- Spindle direction of rotation (**M3**/**M4** miscellaneous function)
- Cycle definition (**CYCL DEF**)

> For some cycles, additional requirements must be observed. They are detailed in the descriptions and overview tables 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 **9 DWELL TIME**
- Cycle **12 PGM CALL**
- Cycle **13 ORIENTATION**
- Cycle **220 POLAR PATTERN**
- Cycle **221 CARTESIAN PATTERN**
- Cycles for coordinate transformation
- 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.

Proceed as follows:

- Press the **CYCL CALL** key

- Press the **CYCL CALL M** soft key
- If required, enter an M function (e.g. **M3**, to switch on the spindle)
- Press **END** to end the dialog

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 360

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 machining 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 **M99** in the positioning block
- > The control moves to the last starting point.

  or
- Define a new machining cycle with **CYCL DEF**

> The control does not support **M89** in combination with free programming of contours!

Calling a cycle with SEL CYCLE

With **SEL CYCLE**, you can call any NC program as a machining cycle.

Proceed as follows:

PGM CALL

- Press the **PGM CALL** key

SELECT CYCLE

- Press the **SELECT CYCLE** soft key

SELECT FILE

- Press the **SELECT FILE** soft key
- Select NC program

Calling an NC program as a cycle

CYCL CALL

- Press the **CYCL CALL** key
- Press the soft key for the cycle call

  or
- Program **M99**

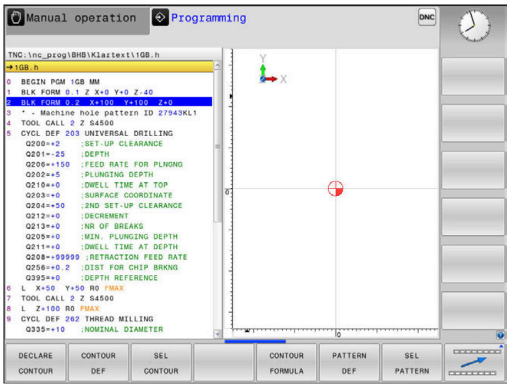
> Programming and operating note:
>
> - If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.
> - When you execute an NC program selected with **SELECT CYCLE**, it will be executed in the Program Run, Single Block operating mode without stopping after each NC block. In addition, it is visible as a single NC block in the Program Run, Full Sequence operating mode.
> - Please note that **CYCL CALL PAT** and **CYCL CALL POS** use a positioning logic before executing the cycle. With respect to the positioning logic, **SELECT CYCLE** and Cycle **12 PGM CALL** show the same behavior: In point pattern cycles, the clearance height is calculated based on the maximum value of all Z positions existing at the starting point of the pattern and all Z positions in the point pattern. With **CYCL CALL POS**, there will be no pre-positioning in the tool axis direction. This means that you need to manually program any pre-positioning in the file you call.

12.4 Program defaults for cycles

Overview

Some cycles always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** function you can define these cycle parameters at the beginning of the program, so that they are effective globally for all cycles used in the NC program. In the respective cycle you then simply reference the value defined at the beginning of the program.

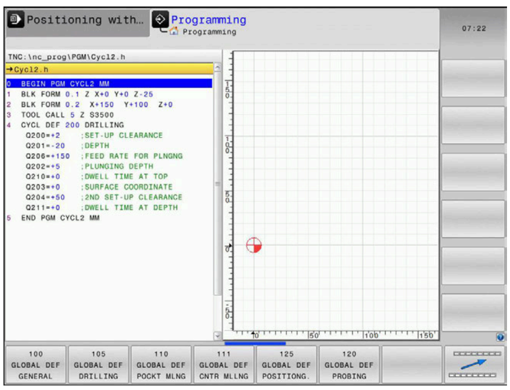
The following **GLOBAL DEF** functions are available:



Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	**GLOBAL DEF** GENERAL Definition of generally valid cycle parameters	356
105 GLOBAL DEF DRILLING	**GLOBAL DEF** DRILLING Definition of specific drilling cycle parameters	357
110 GLOBAL DEF POCKT MLNG	**GLOBAL DEF** POCKET MILLING Definition of specific pocket-milling cycle parameters	358
111 GLOBAL DEF CNTR MLLNG	**GLOBAL DEF** CONTOUR MILLING Definition of specific contour milling cycle parameters	358
125 GLOBAL DEF POSITIONG.	**GLOBAL DEF** POSITIONING Definition of the positioning behavior with **CYCL CALL PAT**	359
120 GLOBAL DEF PROBING	**GLOBAL DEF** PROBING Definition of specific touch probe cycle parameters	359

Entering GLOBAL DEF

Proceed as follows:



- Press the **Programming** key

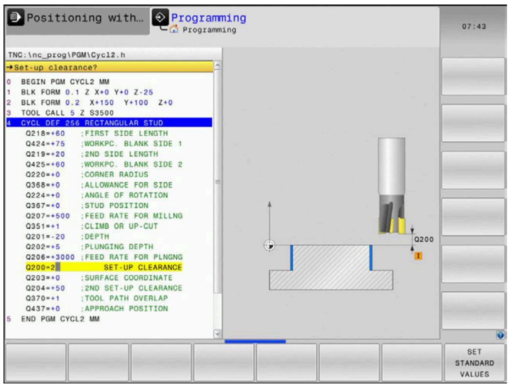
SPEC FCT
- Press the **SPEC FCT** key

PROGRAM DEFAULTS
- Press the **PROGRAM DEFAULTS** soft key

GLOBAL DEF
- Press the **GLOBAL DEF** soft key

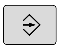
100 GLOBAL DEF GENERAL
- Select the desired **GLOBAL DEF** function (e.g., by pressing the **GLOBAL DEF GENERAL** soft key)
- Enter the required definitions
- Press the **ENT** key each time to confirm

Using GLOBAL DEF information



If you entered the corresponding **GLOBAL DEF** functions at program start, you can reference these globally valid values for the definition of any cycle.

Proceed as follows:



- Press the **PROGRAMMING** key

- Press the **CYCL DEF** key

- Select the desired cycle group (e.g., pockets / studs / slot cycles)

- Select the desired cycle (e.g., **RECTANGULAR STUD**)
- > 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** 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. There is a danger of collision!

- Make sure to use **GLOBAL DEF** carefully. Test your program before executing it
- If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

Global data valid everywhere

The parameters are valid for all **2xx** machining cycles

Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between the tool and the workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Feed rate at which the control moves the tool within a cycle. Input: **0...99999.999** or **FMAX**, **FAUTO**
	Q208 Feed rate for retraction? Feed rate at which the control retracts the tool. Input: **0...99999.999** or **FMAX**, **FAUTO**

Example

```
11 GLOBAL DEF 100 GENERAL ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q208=+999               ;RETRACTION FEED RATE
```

Global data for drilling operations

The parameters apply to the drilling, tapping, and thread milling cycles **200** to **207**, **240**, and **241**.

Help graphic	Parameter
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0.1...99999.9999**
	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. Input: **0...3600.0000**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000**

Example

```
11 GLOBAL DEF 105 DRILLING ~
    Q256=+0.2           ;DIST FOR CHIP BRKNG ~
    Q210=+0             ;DWELL TIME AT TOP ~
    Q211=+0             ;DWELL TIME AT DEPTH
```

Global data for milling operations with pocket cycles

The parameters apply to the cycles **233**, **251**, **253**, and **256**

Help graphic	Parameter
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.1...1999**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (If you enter 0, climb milling is performed.) Input: **-1**, **0**, **+1**
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0**: Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table. **1**: Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message **2**: Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. The reciprocation length depends on the plunging angle. As a minimum value the control uses twice the tool diameter. Input: **0**, **1**, **2**

Example

```
11 GLOBAL DEF 110 POCKET MILLING ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q366=+1                 ;PLUNGE
```

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

The parameters apply to each fixed cycle that you call with the **CYCL CALL PAT** function.

Help graphic	Parameter
	Q345 Select positioning height (0/1) 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. Input: **0**, **1**

Example

```
11 GLOBAL DEF 125 POSITIONING ~
    Q345=+1                 ;SELECT POS. HEIGHT
```

Global data for probing functions

The parameters are valid for all touch probe cycles **4xx**

Help graphic	Parameter
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q301 Move to clearance height (0/1)? Define how the touch probe will move between the measuring points: **0**: Move to measuring height between measuring points **1**: Move to clearance height between measuring points Input: **0**, **1**

Example

```
11 GLOBAL DEF 120 PROBING ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q301=+1                 ;MOVE TO CLEARANCE
```

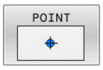
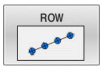
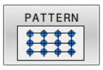
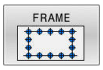
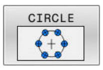
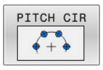
12.5 Pattern definition with PATTERN DEF

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 tool 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	362
	ROW Definition of a single row, straight or rotated	363
	PATTERN Definition of a single pattern, straight, rotated or distorted	364
	FRAME Definition of a single frame, straight, rotated or distorted	366
	CIRCLE Definition of a full circle	368
	PITCH CIRCLE Definition of a pitch circle	369

Entering PATTERN DEF

Proceed as follows:

- Press the **PROGRAMMING** key
- Press the **SPEC FCT** key
- Press the **CONTOUR + POINT MACHINING** soft key
- Press the **PATTERN DEF** soft key
- Select the desired machining pattern (e.g., press the "single row" soft key)
- Enter the required definitions
- Press the **ENT** key each time to confirm

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 351

The control performs the most recently defined machining cycle on the machining pattern you defined.

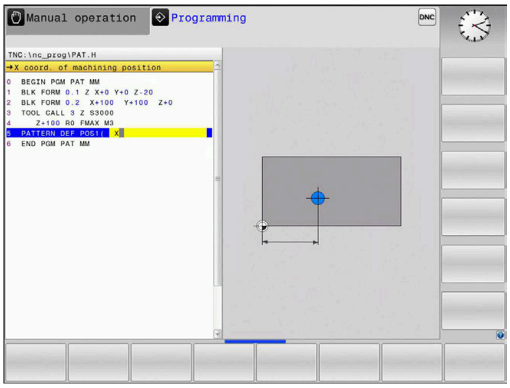
> Programming and operating note:
> - A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.
> - The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the tool axis position 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 set-up clearance and the 2nd set-up clearance reference the coordinate surface in PATTERN DEF.
> - Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q345**=1. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.

> Operating note:
> - You can use the mid-program startup function to select any point at which you want to start or continue machining

Defining individual machining positions

Programming and operating notes:

- 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 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.



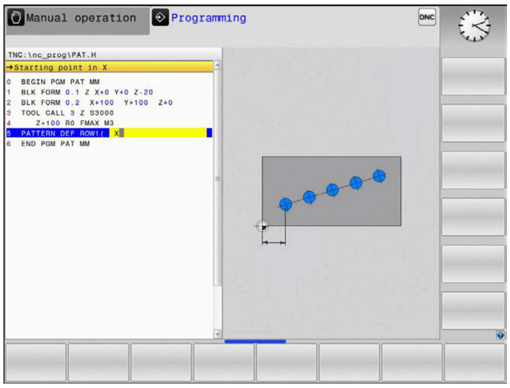
Help graphic	Parameter
	POS1: **X coord. of machining position** Enter the X coordinate as an absolute value. Input: **-999999999...+999999999**
	POS1: **Y coord. of machining position** Enter the Y coordinate as an absolute value. Input: **-999999999...+999999999**
	POS1: **Coordinate of workpiece surface** Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**
	POS2: **X coord. of machining position** Enter the X coordinate as an incremental or absolute value. Input: **-999999999...+999999999**
	POS2: **Y coord. of machining position** Enter the Y coordinate as an incremental or absolute value. Input: **-999999999...+999999999**
	POS2: **Coordinate of workpiece surface** Enter the Z coordinate as an incremental or absolute value. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
POS1( X+25 Y+33.5 Z+0 ) ~
POS2(  X+15 IY+6.5  Z+0 )
```

Defining a single row

> Programming and operating note:
> - 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.



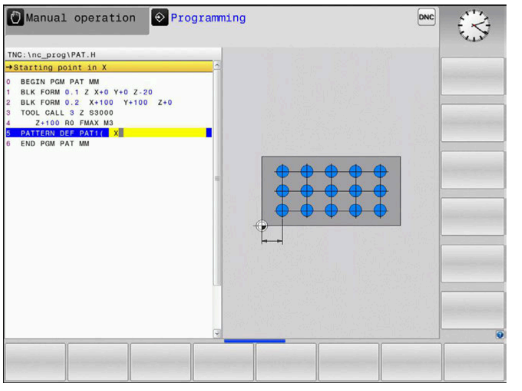
Help graphic	Parameter
	Starting point in X Coordinate of the starting point of the row in the X axis. This value has an absolute effect. Input: **-99999.9999999...+99999.9999999**
	Starting point in Y Coordinate of the starting point of the row in the Y axis. This value has an absolute effect. Input: **-99999.9999999...+99999.9999999**
	Spacing of machining positions Distance (incremental) between the machining positions. Enter a positive or negative value Input: **-999999999...+999999999**
	Number of operations Total number of machining operations Input: **0...999**
	Rot. position of entire pattern Angle of rotation around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
ROW1( X+25 Y+33.5 D+8 NUM5 ROT+0 Z+0 )
```

Defining an individual pattern

> Programming and operating notes:
> - The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire 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.



Help graphic	Parameter
	Starting point in X Absolute coordinate of the pattern starting point in the X axis Input: **-999999999...+999999999**
	Starting point in Y Absolute coordinate of the pattern starting point in the Y axis Input: **-999999999...+999999999**
	Spacing of machining positions X Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Spacing of machining positions Y Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Number of columns Total number of columns in the pattern Input: **0...999**
	Number of rows Total number of rows in the pattern Input: **0...999**
	Rot. position of entire pattern Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Rotary pos. ref. ax. Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value Input: **-360.000...+360.000**

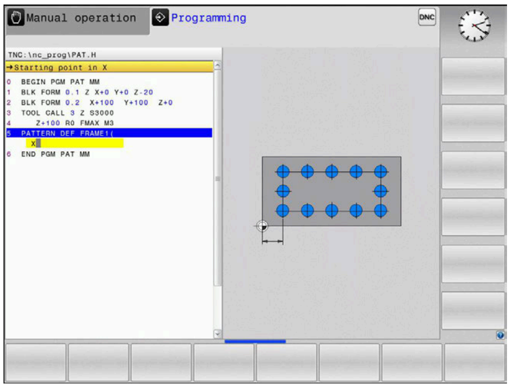
Help graphic	Parameter
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
PAT1( X+25 Y+33.5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0
    ROTY+0 Z+0 )
```

Defining an individual frame

> Programming and operating notes:
> - The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire 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.



Help graphic	Parameter
	Starting point in X Absolute coordinate of the frame starting point in the X axis Input: **-999999999...+999999999**
	Starting point in Y Absolute coordinate of the frame starting point in the Y axis Input: **-999999999...+999999999**
	Spacing of machining positions X Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Spacing of machining positions Y Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Number of columns Total number of columns in the pattern Input: **0...999**
	Number of rows Total number of rows in the pattern Input: **0...999**
	Rot. position of entire pattern Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Rotary pos. ref. ax. Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: **-360.000...+360.000**

Help graphic	Parameter
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts Input: **-999999999...+999999999**

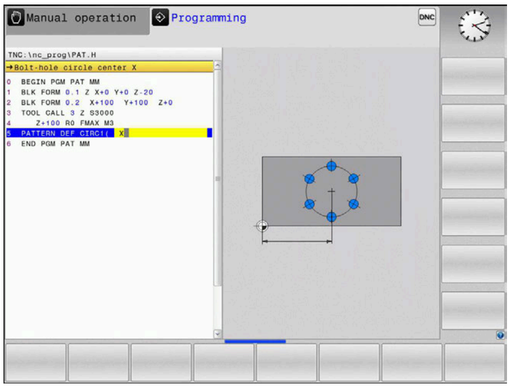
Example

```
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

Programming and operating notes:

- 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.



Help graphic	Parameter
	Bolt-hole circle center X Absolute coordinate of the circle center point in the X axis Input: **-999999999...+999999999**
	Bolt-hole circle center Y Absolute coordinate of the circle center point in the Y axis Input: **-999999999...+999999999**
	Bolt-hole circle diameter Diameter of the bolt hole circle Input: **0...999999999**
	Starting angle Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). You can enter a positive or negative value Input: **-360.000...+360.000**
	Number of operations Total number of machining positions on the circle Input: **0...999**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**

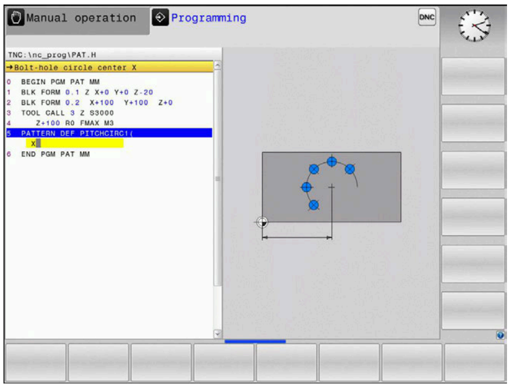
Example

```
11 PATTERN DEF ~
CIRC1( X+25 Y+33 D80 START+45 NUM8 Z+0 )
```

Defining a pitch circle

Programming and operating notes:

- 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.



Help graphic	Parameter
	Bolt-hole circle center X Absolute coordinate of the circle center point in the X axis Input: **-999999999...+999999999**
	Bolt-hole circle center Y Absolute coordinate of the circle center point in the Y axis Input: **-999999999...+999999999**
	Bolt-hole circle diameter Diameter of the bolt hole circle Input: **0...999999999**
	Starting angle Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). You can enter a positive or negative value Input: **-360.000...+360.000**
	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 Stopping angle (switch via soft key) Input: **-360.000...+360.000**
	Number of operations Total number of machining positions on the circle Input: **0...999**
	Coordinate of workpiece surface Enter the Z coordinate at which machining starts. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
PITCHCIRC1( X+25 Y+33 D80 START+45 STEP+30 NUM8 Z+0 )
```

12.6 Cycle 220 POLAR PATTERN

Application

This cycle enables you to define a point pattern as a full or pitch circle. It can be used for a previously defined machining cycle.

Related topics

- Defining a full circle with **PATTERN DEF**

  Further information: "Defining a full circle", Page 368
- Defining a circle segment with **PATTERN DEF**

  Further information: "Defining a pitch circle", Page 369

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 2nd set-up clearance (spindle axis)
  - Approach the starting point in the working plane
  - Move to 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 procedure (steps 1 to 3) will be repeated until all machining operations have been completed

> If you run this cycle in Single Block mode, the control stops between the individual points of a point pattern.

Notes

> Cycle **220 POLAR PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

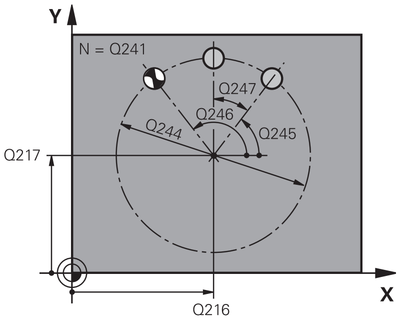
- Cycle **220** is DEF-active. In addition, Cycle **220** automatically calls the last defined machining cycle.

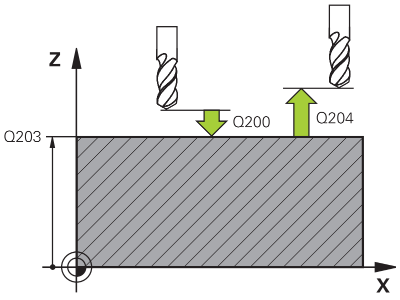
Note on programming

- If you combine one of the machining cycles **200** to **207** or **251**, **253** or **256** with Cycle **220** or Cycle **221**, the set-up clearance, the workpiece surface, and the 2nd set-up clearance from Cycle **220** or **221** are effective. This applies within the NC program until the affected parameters are overwritten again.

  Example: If Cycle **200** is defined in an NC program with **Q203**=0 and you then program Cycle **220** with **Q203**=-5, then the subsequent calls 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).

Cycle parameters

Help graphic	Parameter
	**Q216 Center in 1st axis?** Pitch circle center in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q217 Center in 2nd axis? Pitch circle center in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q244 Pitch circle diameter? Diameter of circle Input: **0...99999.9999**
	Q245 Starting angle? Angle between the main axis of the working plane and the starting point for the first machining operation on the pitch circle. This value has an absolute effect. Input: **-360.000...+360.000**
	Q246 Stopping angle? Angle between the main 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. This value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the control will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the control will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). This value has an incremental effect. Input: **-360.000...+360.000**
	Q241 Number of repetitions? Number of machining operations on a pitch circle Input: **1...99999**

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q301 Move to clearance height (0/1)? Specify how the tool moves between machining processes: **0**: Move to the set-up clearance between operations **1**: Move to the 2nd set-up clearance between operations Input: **0**, **1**

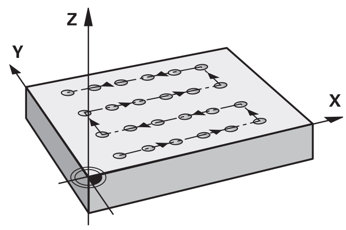
Example

```
11 CYCL DEF 220 POLAR PATTERN ~
    Q216=+50              ;CENTER IN 1ST AXIS ~
    Q217=+50              ;CENTER IN 2ND AXIS ~
    Q244=+60              ;PITCH CIRCLE DIAMETR ~
    Q245=+0               ;STARTING ANGLE ~
    Q246=+360             ;STOPPING ANGLE ~
    Q247=+0               ;STEPPING ANGLE ~
    Q241=+8               ;NR OF REPETITIONS ~
    Q200=+2               ;SET-UP CLEARANCE ~
    Q203=+30              ;SURFACE COORDINATE ~
    Q204=+50              ;2ND SET-UP CLEARANCE ~
    Q301=+1               ;MOVE TO CLEARANCE
12 CYCL CALL
```

12.7 Cycle 221 CARTESIAN PATTERN

Application

This cycle enables you to define a point pattern as lines. It can be used for a previously defined machining cycle.



Related topics

- Defining an individual row with **PATTERN DEF**
  Further information: "Defining a single row", Page 363
- Defining an individual pattern with **PATTERN DEF**
  Further information: "Defining an individual pattern", Page 364

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 2nd set-up clearance (spindle axis)
  - Approach the starting point in the working plane
  - Move to 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 procedure (step 6) will be repeated until all machining operations of the second line have been completed
8 The tool then moves to the starting point of the next row
9 All subsequent lines are machined in a reciprocating movement.

> If you run this cycle in Single Block mode, the control stops between the individual points of a point pattern.

Notes

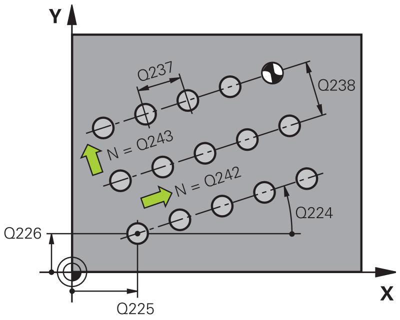
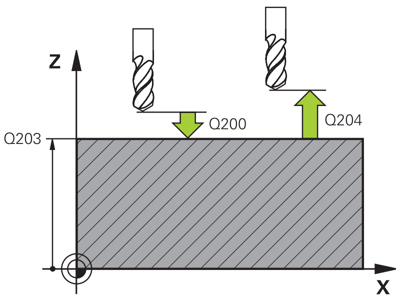
> Cycle **221 CARTESIAN PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined machining cycle.

Notes on programming

- If you combine Cycle **221** with one of the machining cycles **200** to **207** or **251**, **253** and **256**, then the set-up clearance, the workpiece surface, the 2nd set-up clearance, and the rotary position that you defined in Cycle **221** will be effective for the selected machining cycle.

Cycle parameters

Help graphic	Parameter
 	**Q225 Starting point in 1st axis?** Coordinate of starting point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Coordinate of starting point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q237 Spacing in 1st axis? Spacing between the individual points on a line. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q238 Spacing in 2nd axis? Spacing between the individual lines. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q242 Number of columns? Number of machining operations on a line Input: **0...99999**
	Q243 Number of lines? Number of lines Input: **0...99999**
	Q224 Angle of rotation? Angle by which the entire pattern is rotated. The center of rotation lies in the starting point. This value has an absolute effect. Input: **-360.000...+360.000**
 	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the tool moves between machining processes: **0**: Move to the set-up clearance between operations **1**: Move to the 2nd set-up clearance between operations Input: **0**, **1**

Example

```
11 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=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q301=+1             ;MOVE TO CLEARANCE
12 CYCL CALL
```

12.8 Point tables with cycles

Application with cycles

With a point table you can execute one or more cycles in sequence on an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting point coordinates of the respective cycle . Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Related topics

- Contents of a point table, hiding individual points
  Further information: "Point tables", Page 188

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**:

Proceed as follows:

CYCL CALL

- Press the **CYCL CALL** key

CYCLE CALL PAT

- Press the **CYCL CALL PAT** soft key
- Enter a feed rate
  or
- Press the **F MAX** soft key
- > The control will use this feed rate to traverse between the points.
- > No input: the control will use 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 (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q345**=1. If you do so, the control will always position the tool at the 2nd 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.

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! There is a danger of collision!

- 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.

Programming and operating notes:

- 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**.

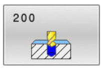
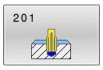
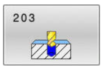
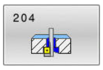
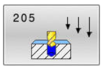
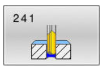
13

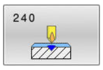
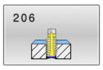
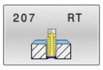
Cycles: Drilling Cycles / Thread Cycles

13.1 Fundamentals

Overview

The control provides the following cycles for all types of drilling and threading operations:

Soft key	Cycle	Page
	Cycle 200 DRILLING ■ Basic hole ■ Input of the dwell time at top and bottom ■ Depth reference selectable	385
	Cycle 201 REAMING ■ Reaming a hole ■ Input of the dwell time at bottom	389
	Cycle 202 REAMING ■ Boring a hole ■ Input of the retraction feed rate ■ Input of the dwell time at bottom ■ Input of the retracting movement	391
	Cycle 203 UNIVERSAL DRILLING ■ Degression – hole with decreasing infeed ■ Input of the dwell time at top and bottom ■ Input of chip breaking behavior ■ Depth reference selectable	395
	Cycle 204 BACK BORING ■ Machining a counterbore on the underside of the workpiece ■ Input of the dwell time ■ Input of the retracting movement	401
	Cycle 205 UNIVERSAL PECKING ■ Degression – hole with decreasing infeed ■ Input of chip breaking behavior ■ Input of a deepened starting point ■ Input of an advanced stop distance	405
	Cycle 241 SINGLE-LIP D.H.DRLNG ■ Drilling with single-lip deep hole drill ■ Deepened starting point ■ Direction of rotation and rotational speed for moving into and retracting from the hole ■ Input of the dwell depth	413

Soft key	Cycle	Page
	Cycle 240 CENTERING ■ Drilling a center hole ■ Input of the centering diameter or depth ■ Input of the dwell time at bottom	382
	Cycle 206 TAPPING ■ Tapping with floating tap holder ■ Input of the dwell time at bottom	426
	Cycle 207 RIGID TAPPING ■ Rigid tapping ■ Input of the dwell time at bottom	429

13.2 Cycle 240 CENTERING

Application

Use Cycle **240 CENTERING** to machine center holes. You can specify the centering diameter or depth and an optional dwell time at the bottom. This dwell time is used for chip breaking at the bottom of the hole. If there is already a pilot hole then you can enter a deepened starting point.

Cycle sequence

1. From the current position, the control positions the tool at rapid traverse **FMAX** in the working plane to the starting position.
2. The control positions the tool at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** above the workpiece surface **Q203**.
3. If you define **Q342 ROUGHING DIAMETER** not equal to 0, the control uses this value and the point angle of the tool **T-ANGLE** to calculate a deepened starting point. The control positions the tool at the **F PRE-POSITIONING Q253** feed rate to the deepened starting point.
4. The tool is centered at the programmed feed rate for plunging **F** to the programmed centering diameter or centering depth.
5. If a dwell time **Q211** is defined, the tool remains at the centering depth.
6. Finally, the tool is retracted to the set-up clearance or to the 2nd set-up 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**.

Notes

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! There is a danger of collision!

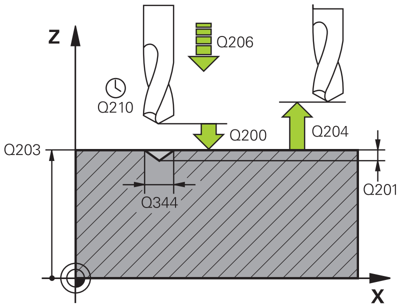
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

Notes on programming

- Program a positioning block to position the tool at the starting point (hole center) in the working plane with 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.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...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 Input: **0**, **1**
	Q201 Depth? Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if **Q343**=0 is defined. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q344 Diameter of counterbore Centering diameter. Only effective if **Q343**=1 is defined. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while centering Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q342 Roughing diameter? **0**: There is no hole **>0**: Diameter of the pre-drilled hole Input: **0...99999.9999**

Help graphic	Parameter
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the deepened starting point. The speed is in mm/min. Only in effect if **Q342 ROUGHING DIAMETER** is not 0. Input: **0...99999.9999** or **FMAX**, **FAUTO**

Example

```
11 CYCL DEF 240 CENTERING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q343=+1              ;SELECT DIA./DEPTH ~
    Q201=-2              ;DEPTH ~
    Q344=-10             ;DIAMETER ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q211=+0              ;DWELL TIME AT DEPTH ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q342=+12             ;ROUGHING DIAMETER ~
    Q253=+500            ;F PRE-POSITIONING
12 L  X+30 R0 FMAX
13 L  Y+20 R0 FMAX M3 M99
14 L  X+80 R0 FMAX
15 L  X+50 R0 FMAX M99
```

13.3 Cycle 200 DRILLING

Application

With this cycle, you can drill basic holes. In this cycle, the depth reference is selectable.

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 set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to 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 procedure (steps 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 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**

Notes

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! There is a danger of collision!

- 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

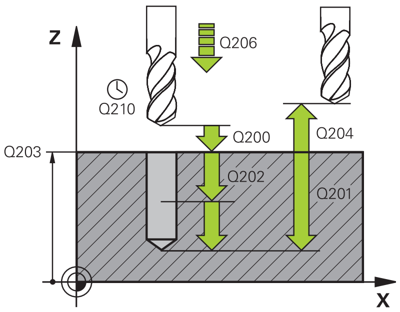
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

> 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.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q202 Plunging depth? Tool infeed per cut. This value has an incremental effect. 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 Input: **0...99999.9999**
	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. Input: **0...3600.0000**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active preset. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000**

Help graphic	Parameter
	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 table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**

Example

```
11 CYCL DEF 200 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
12 L  X+30 FMAX
13 L  Y+20 FMAX M3 M99
14 L  X+80 FMAX
15 L  Y+50 FMAX M99
```

13.4 Cycle 201 REAMING

Application

With this cycle, you can machine basic fits. In this cycle, you can optionally define a dwell time at the bottom of the hole.

Cycle sequence

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 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**

Notes

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! There is a danger of collision!

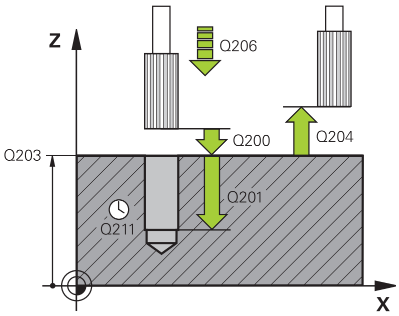
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on 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.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while reaming Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...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: **0...99999.9999** or **FMAX**, **FAUTO**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active preset. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**

Example

```
11 CYCL DEF 201 REAMING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q201=-20             ;DEPTH ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q211=+0              ;DWELL TIME AT DEPTH ~
    Q208=+99999          ;RETRACTION FEED RATE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE
12 L  X+30 FMAX
13 L  Y+20 FMAX M3 M99
```

13.5 Cycle 202 REAMING

Application

Refer to your machine manual.
This cycle is effective only for machines with servo-controlled spindle.

With this cycle, you can bore holes. In this cycle, you can optionally define a dwell time at the bottom of the hole.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the safety clearance **Q200** above the workpiece **Q203 SURFACE COORDINATE**
2 The tool drills to the programmed depth at the feed rate for plunging **Q201**
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 **Q214 DISENGAGING DIRECTN** is defined, the control retracts in the programmed direction by the value in **CLEARANCE TO SIDE Q357**
6 Then the control moves the tool at the retraction feed rate **Q208** to the set-up clearance **Q200**
7 The tool is again centered in the hole
8 The control restores the spindle status as it was at the cycle start.
9 If programmed, the control moves the tool at **FMAX** to 2nd set-up clearance. 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

Notes

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! There is a danger of collision!

- 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 programming an oriented spindle stop with reference to the angle entered in **Q336** (e.g., in the **Positioning w/ Manual Data Input** operating mode). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

NOTICE

Danger of collision!

If you have activated **M136**, the tool will not move to the programmed set-up clearance once the machining operation is finished. The spindle rotation will stop at the bottom of the hole which, in turn, also stops the feed motion. There is a danger of collision as the tool will not be retracted!

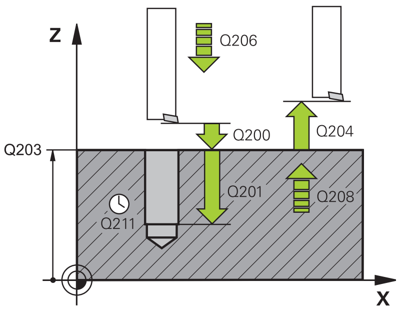
- Use **M137** to deactivate **M136** before the cycle start

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- If **Q214 DISENGAGING DIRECTN** is not 0, **Q357 CLEARANCE TO SIDE** is in effect.

Notes on 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.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while boring Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...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: **0...99999.9999** or **FMAX**, **FAUTO**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q214 Disengaging directn (0/1/2/3/4)? Specify the direction in which the control retracts the tool at the hole bottom (after carrying out an oriented spindle stop) **0**: Do not retract tool **1**: Retract tool in negative main axis direction **2**: Retract tool in negative secondary axis direction **3**: Retract tool in positive main axis direction **4**: Retract tool in positive secondary axis direction Input: **0**, **1**, **2**, **3**, **4**
	Q336 Angle for spindle orientation? Angle to which the control positions the tool before retracting it. This value has an absolute effect. Input: **0...360**

Help graphic	Parameter
	Q357 Safety clearance to the side? Distance between tool tooth and the wall. This value has an incremental effect. Only in effect if **Q214 DISENGAGING DIRECTN** is not 0. Input: **0...99999.9999**

Example

```
11 L  Z+100 R0 FMAX
12 CYCL DEF 202 BORING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q201=-20             ;DEPTH ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q211=+0              ;DWELL TIME AT DEPTH ~
    Q208=+99999          ;RETRACTION FEED RATE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q214=+1              ;DISENGAGING DIRECTN ~
    Q336=+0              ;ANGLE OF SPINDLE ~
    Q357=+0.2            ;CLEARANCE TO SIDE
13 L  X+30 FMAX
14 L  Y+20 FMAX M3 M99
15 L  X+80 FMAX
16 L  Y+50 FMAX M99
```

13.6 Cycle 203 UNIVERSAL DRILLING

Application

With this cycle, you can drill holes with decreasing infeed. In this cycle, you can optionally define a dwell time at the bottom of the hole. The cycle may be executed with or without chip breaking.

Related topics

- Cycle **200 DRILLING** for simple holes

  Further information: "Cycle 200 DRILLING", Page 385
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance

  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 405
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole

  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 413

Cycle run

Behavior without chip breaking, 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 DEPTH Q202** at the **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, if necessary, remains there for the **DWELL TIME AT TOP Q210**
6. This sequence will be repeated until the **DEPTH Q201** is reached.
7. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **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, 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 the **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 the **SET-UP CLEARANCE Q200** or to the **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, with 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** 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 the **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 the **SET-UP CLEARANCE Q200** or to the **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**

Notes

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! There is a danger of collision!

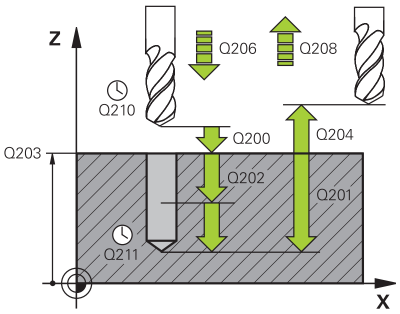
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on 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.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q202 Plunging depth? Tool infeed per cut. This value has an incremental effect. 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 Input: **0...99999.9999**
	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. Input: **0...3600.0000**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q212 Decrement? Value by which the control decreases **Q202 PLUNGING DEPTH** after each infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q213 Nr of breaks before retracting? Number of chip breaks after which the control is to withdraw the tool from the hole for chip breaking. For chip breaking, the control retracts the tool each time by the value in **Q256**. Input: **0...99999**

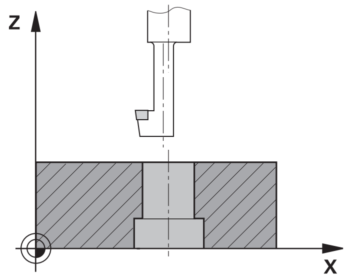
Help graphic	Parameter
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...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: **0...99999.9999** or **FMAX**, **FAUTO**
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...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 table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**

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 ~
    Q210=+0                 ;DWELL TIME AT TOP ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q212=+0                 ;DECREMENT ~
    Q213=+0                 ;NR OF BREAKS ~
    Q205=+0                 ;MIN. PLUNGING DEPTH ~
    Q211=+0                 ;DWELL TIME AT DEPTH ~
    Q208=+99999             ;RETRACTION FEED RATE ~
    Q256=+0.2               ;DIST FOR CHIP BRKNG ~
    Q395=+0                 ;DEPTH REFERENCE
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
```

13.7 Cycle 204 BACK BORING

Application



Refer to your machine manual. This cycle is effective only for machines with servo-controlled spindle.

Special boring bars for upward cutting are required for this cycle.

This cycle allows counterbores to be machined from the underside of the workpiece.

Cycle sequence

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 the 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, if applicable, the coolant and moves the tool 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 moves at **FMAX** to set-up clearance.
7 The tool is again centered in the hole
8 The control restores the spindle status as it was at the cycle start.
9 If necessary, the control moves the tool to 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Notes

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 programming an oriented spindle stop with reference to the angle entered in **Q336** (e.g., in the **Positioning w/ Manual Data Input** operating mode). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

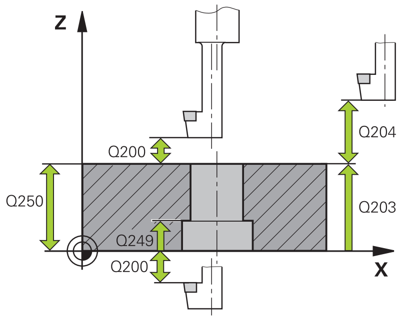
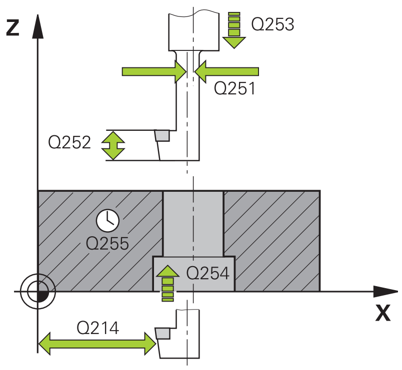
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- When calculating the starting point for boring, the control considers 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.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF COUNTERBORE Q249**, the control will display an error message.

Enter the tool length measured up to the lower edge of the boring bar, not the cutting edge.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the cycle parameter depth determines the working direction. Note: If you enter a positive sign, the tool bores in the direction of the positive spindle axis.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q249 Depth of counterbore? Distance between underside of workpiece and the top of hole. A positive sign means the hole will be bored in the positive spindle axis direction. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q250 Material thickness? Height of the workpiece. Enter an incremental value. Input: **0.0001...99999.9999**
	**Q251 Tool edge off-center distance?** Off-center distance of the boring bar. Refer to the tool data sheet. This value has an incremental effect. Input: **0.0001...99999.9999**
	Q252 Tool edge height? Distance between underside of boring bar and main cutting tooth. Refer to the tool data sheet. This value has an incremental effect.
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q255 Dwell time in secs.? Dwell time in seconds at the bottom of the bore hole Input: **0...99999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q214 Disengaging directn (0/1/2/3/4)? Specify the direction in which the control offsets the tool by the off-center distance (after orienting the spindle). Inputting 0 is not permitted **1**: Retract tool in negative main axis direction **2**: Retract tool in negative secondary axis direction **3**: Retract tool in positive main axis direction **4**: Retract tool in positive secondary axis direction Input: **1**, **2**, **3**, **4**
	Q336 Angle for spindle orientation? Angle at which the control positions the tool before it is plunged into or retracted from the bore hole This value has an absolute effect. Input: **0...360**

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 ~
    Q254=+200           ;F COUNTERBORING ~
    Q255=+0             ;DWELL TIME ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q214=+0             ;DISENGAGING DIRECTN ~
    Q336=+0             ;ANGLE OF SPINDLE
12 CYCL CALL
```

13.8 Cycle 205 UNIVERSAL PECKING

Application

With this cycle, you can drill holes with decreasing infeed. The cycle may be executed with or without chip breaking. When the plunging depth is reached the cycle performs chip removal. If there is already a pilot hole then you can enter a deepened starting point. In this cycle, you can optionally define a dwell time at the bottom of the hole. This dwell time is used for chip breaking at the bottom of the hole.

Further information: "Chip removal and chip breaking", Page 411

Related topics

- Cycle **200 DRILLING** for simple holes

  Further information: "Cycle 200 DRILLING", Page 385
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking

  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 395
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole

  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 413

Cycle run

1. The control positions the tool in the tool axis at **FMAX** to the entered **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.
2. If you program a deepened starting point in **Q379**, the control moves at the positioning feed rate **Q253 F PRE-POSITIONING** to the set-up clearance above the deepened starting point.
3. The tool drills at the programmed **Q206 FEED RATE FOR PLNGNG** to the plunging depth.
4. If you have programmed chip breaking, the control retracts the tool by the retraction value **Q256**.
5. Upon reaching the plunging depth, the control retracts the tool in the tool axis at the retraction feed rate **Q208** to the set-up clearance. The set-up clearance is above the **SURFACE COORDINATE Q203**.
6. The tool then moves at **FMAX** to the entered advanced stop distance above the plunging depth last reached.
7. The tool drills at the feed in **Q206** to the next plunging depth. If a decrement Q212 is defined, the plunging depth is decreased after each infeed by the decrement.
8. The control repeats this procedure (steps 2 to 7) until the total drilling depth is reached.
9. If you entered a dwell time, the tool remains at the hole bottom for chip breaking. The control then retracts the tool at the retraction feed rate to the set-up clearance or the 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

After chip removal, the depth of the next chip breaking is referenced to the last plunging depth.

Example:

- **Q202 PLUNGING DEPTH** = 10 mm
- **Q257 DEPTH FOR CHIP BRKNG** = 4 mm

  The control performs chip breaking at 4 mm and 8 mm. Chip removal is performed at 10 mm. Chip breaking is next performed at 14 mm and 18 mm, etc.

Notes

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! There is a danger of collision!

- 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

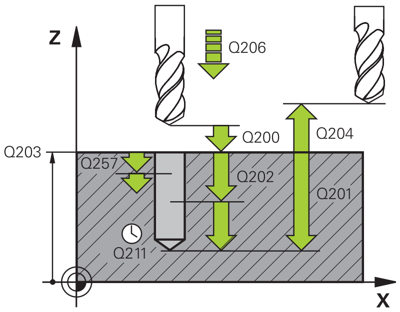
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

This cycle is not suitable for overlong drills. For overlong drills, use Cycle **241 SINGLE-LIP D.H.DRLNG**.

Notes on 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 deepened 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.
- If **Q257 DEPTH FOR CHIP BRKNG** is greater than **Q202 PLUNGING DEPTH**, the operation is executed without chip breaking.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between workpiece surface and bottom of hole (depends on parameter **Q395 DEPTH REFERENCE**). This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q202 Plunging depth? Tool infeed per cut. This value has an incremental effect. 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 Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q212 Decrement? Value by which the control decreases the plunging depth **Q202**. This value has an incremental effect. Input: **0...99999.9999**
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q258 Upper advanced stop distance? Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after first chip removal. This value has an incremental effect. Input: **0...99999.9999**
	Q259 Lower advanced stop distance? Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after the last chip removal. This value has an incremental effect. Input: **0...99999.9999**
	Q257 Infeed depth for chip breaking? Incremental depth at which the control performs chip breaking. This procedure is repeated until **DEPTH Q201** is reached. If **Q257** equals 0, the control will not perform chip breaking. This value has an incremental effect. Input: **0...99999.9999**
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...99999.999**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000**
	Q379 Deepened starting point? If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Defines the tool traversing speed when positioning from **Q200 SET-UP CLEARANCE** to **Q379 STARTING POINT** (not equal to 0). Input in mm/min. Input: **0...99999.9999** or **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: **0...99999.9999** or **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 table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**

Example

```
11 CYCL DEF 205 UNIVERSAL PECKING ~
    Q200=+2                  ;SET-UP CLEARANCE ~
    Q201=-20                 ;DEPTH ~
    Q206=+150                ;FEED RATE FOR PLNGNG ~
    Q202=+5                  ;PLUNGING DEPTH ~
    Q203=+0                  ;SURFACE COORDINATE ~
    Q204=+50                 ;2ND SET-UP CLEARANCE ~
    Q212=+0                  ;DECREMENT ~
    Q205=+0                  ;MIN. PLUNGING DEPTH ~
    Q258=+0.2                ;UPPER ADV STOP DIST ~
    Q259=+0.2                ;LOWER ADV STOP DIST ~
    Q257=+0                  ;DEPTH FOR CHIP BRKNG ~
    Q256=+0.2                ;DIST FOR CHIP BRKNG ~
    Q211=+0                  ;DWELL TIME AT DEPTH ~
    Q379=+0                  ;STARTING POINT ~
    Q253=+750                ;F PRE-POSITIONING ~
    Q208=+99999              ;RETRACTION FEED RATE ~
    Q395=+0                  ;DEPTH REFERENCE ~
```

Chip removal and chip breaking

Chip removal

Chip removal depends on cycle parameter **Q202 PLUNGING DEPTH**.

When the value entered in cycle parameter **Q202** is reached, the control performs chip removal. This means that the control always moves the tool to the retraction height, irrespective of the deepened starting point **Q379**. This height is calculated from **Q200 SET-UP CLEARANCE** + **Q203 SURFACE COORDINATE**

Example:

0 BEGIN PGM 205 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 203 Z S4500	; Tool call (tool radius 3)
4 L Z+250 R0 FMAX	; Retract the tool
5 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q212=+0 ;DECREMENT ~	
Q205=+0 ;MIN. PLUNGING DEPTH ~	
Q258=+0.2 ;UPPER ADV STOP DIST ~	
Q259=+0.2 ;LOWER ADV STOP DIST ~	
Q257=+0 ;DEPTH FOR CHIP BRKNG ~	
Q256=+0.2 ;DIST FOR CHIP BRKNG ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q379=+10 ;STARTING POINT ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q208=+3000 ;RETRACTION FEED RATE ~	
Q395=+0 ;DEPTH REFERENCE	
6 L X+30 R0 FMAX M3	; Approach drilling position in the X axis, spindle ON
7 L Y+30 R0 FMAX M3	; Approach drilling position in the Y axis
8 CYCL CALL	; Cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 M30	; End of program
11 END PGM 205 MM	

Chip breaking

Chip breaking depends on cycle parameter **Q257 DEPTH FOR CHIP BRKNG**.

When the value entered in cycle parameter **Q257** is reached, the control performs chip breaking. This means that the control retracts the tool by the value defined in **Q256 DIST FOR CHIP BRKNG**. Chip removal starts once the tool reaches the **PLUNGING DEPTH**. The entire process is repeated until **Q201 DEPTH** is reached.

Example:

0 BEGIN PGM 205 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 203 Z S4500	; Tool call (tool radius 3)
4 L Z+250 R0 FMAX	; Retract the tool
5 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q212=+0 ;DECREMENT ~	
Q205=+0 ;MIN. PLUNGING DEPTH ~	
Q258=+0.2 ;UPPER ADV STOP DIST ~	
Q259=+0.2 ;LOWER ADV STOP DIST ~	
Q257=+3 ;DEPTH FOR CHIP BRKNG ~	
Q256=+0.5 ;DIST FOR CHIP BRKNG ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q379=+0 ;STARTING POINT ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q208=+3000 ;RETRACTION FEED RATE ~	
Q395=+0 ;DEPTH REFERENCE	
6 L X+30 R0 FMAX M3	; Approach drilling position in the X axis, spindle ON
7 L Y+30 R0 FMAX M3	; Approach drilling position in the Y axis
8 CYCL CALL	; Cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 M30	; End of program
11 END PGM 205 MM	

13.9 Cycle 241 SINGLE-LIP D.H.DRLNG

Application

Cycle **241 SINGLE-LIP D.H.DRLNG** machines holes with a single-lip deep hole drill. It is possible to enter a recessed starting point. The control performs moving to drilling depth with **M3**. You can change the direction of rotation and the rotational speed for moving into and retracting from the hole.

Related topics

- Cycle **200 DRILLING** for simple holes

  Further information: "Cycle 200 DRILLING", Page 385
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking

  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 395
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance

  Further information: "Cycle 205 UNIVERSAL PECKING ", Page 405

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

   Further information: "Position behavior when working with Q379", Page 419
2 Depending on the "Position behavior when working with Q379", Page 419, 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
3 The control executes the approach motion depending on the definition of **Q426 DIR. OF SPINDLE ROT.** with a spindle that rotates clockwise, counterclockwise, or is stationary
4 The tool drills with **M3** and **Q206 FEED RATE FOR PLNGNG** to the drilling depth **Q201** or dwell depth **Q435** or the plunging depth **Q202**:
   - After defining **Q435 DWELL DEPTH**, the control reduces the feed rate by **Q401 FEED RATE FACTOR** after reaching the dwell depth and remains there for **Q211 DWELL TIME AT DEPTH**
   - If a smaller infeed value has been entered, the control drills to the plunging depth. The plunging depth is decreased after each infeed by **Q212 DECREMENT**
5 If programmed, the tool remains at the hole bottom for chip breaking.
6 After the control has reached the hole depth, it will automatically switch off the coolant, set the speed to the value defined in **Q427 ROT.SPEED INFEED/OUT** and, if required, change again the direction of rotation from **Q426**.
7 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 419
8 If programmed, the tool moves to 2nd set-up clearance at **FMAX**

Notes

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! There is a danger of collision!

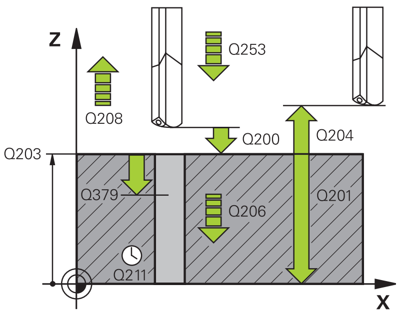
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on 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.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and **Q203 SURFACE COORDINATE**. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between **Q203 SURFACE COORDINATE** and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active preset. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q379 Deepened starting point? If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect. Input: **0...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: **0...99999.9999** or **FMAX**, **FAUTO**

Help graphic	Parameter
	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: **0...99999.999** or **FMAX**, **FAUTO**
	Q426 Rot. dir. of entry/exit (3/4/5)? Rotational speed at which the tool is to rotate when moving into and retracting from the hole. **3**: Spindle rotation with M3 **4**: Spindle rotation with M4 **5**: Movement with stationary spindle Input: **3**, **4**, **5**
	Q427 Spindle speed of entry/exit? Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input: **1...99999**
	Q428 Spindle speed for drilling? Desired speed for drilling. Input: **0...99999**
	Q429 M function for coolant on? **>=0**: Miscellaneous function M for switching on the coolant. The control switches the coolant on when the tool has reached the set-up clearance **Q200** above the starting point **Q379**. **"..."**: Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically. **Further information:** "User macro", Page 418 Input: **0...999**
	Q430 M function for coolant off? **>=0**: Miscellaneous function M for switching off the coolant. The control switches the coolant off if the tool is at **Q201 DEPTH**. **"..."**: Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically. **Further information:** "User macro", Page 418 Input: **0...999**

Help graphic	Parameter
	Q435 Dwell depth? 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**. This value has an incremental effect. Input: **0...99999.9999**
	Q401 Feed rate factor in %? Factor by which the control reduces the feed rate after reaching **Q435 DWELL DEPTH**. Input: **0.0001...100**
	Q202 Maximum plunging depth? Infeed per cut. The **DEPTH Q201** does not have to be a multiple of **Q202**. This value has an incremental effect. Input: **0...99999.9999**
	Q212 Decrement? Value by which the control decreases **Q202 PLUNGING DEPTH** after each infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**

Example

11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q379=+0	;STARTING POINT ~
Q253=+750	;F PRE-POSITIONING ~
Q208=+1000	;RETRACTION FEED RATE ~
Q426=+5	;DIR. OF SPINDLE ROT. ~
Q427=+50	;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=+99999	;MAX. PLUNGING DEPTH ~
Q212=+0	;DECREMENT ~
Q205=+0	;MIN. PLUNGING DEPTH
12 CYCL CALL	

User macro

A user macro is another NC program.

A user macro contains a sequence of multiple instructions. With a macro, you can define multiple NC functions that the control executes. As a user, you create macros as an NC program.

Macros work in the same manner as NC programs that are called with the NC function **CALL PGM**, for example. You define a macro as an NC program with the file type *.h or *.i.

- HEIDENHAIN recommends using QL parameters in the macro. QL parameters have only a local effect for an NC program. If you use other types of variables in the macro, then changes may also have an effect on the calling NC program. In order to explicitly cause changes in the calling NC program, use Q or QS parameters with the numbers 1200 to 1399.
- Within the macro, you can read the value of the cycle parameters.

  Further information: Klartext Programming User's Manual

Example of a user macro for coolant

0 BEGIN PGM KM MM	
1 FN 18: SYSREAD QL100 = ID20 NR8	; Read the coolant level
2 FN 9: IF QL100 EQU +1 GOTO LBL "Start"	; Query the coolant level; if coolant is active, jump to the **Start** LBL
3 M8	; Switch coolant on
7 CYCL DEF 9.0 DWELL TIME	
8 CYCL DEF 9.1 V.ZEIT3	
9 LBL "Start"	
10 END PGM RET MM	

Position behavior when working with Q379

Especially when working with very long drills (for example, 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 deepened 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 of drilling is 0.4 mm or 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, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (**Q200**=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (**Q200**=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

The point at which the 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 deepened 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 or inches above the recessed start point. So if the recessed starting point is at –2, the control starts chip removal at –0.4.

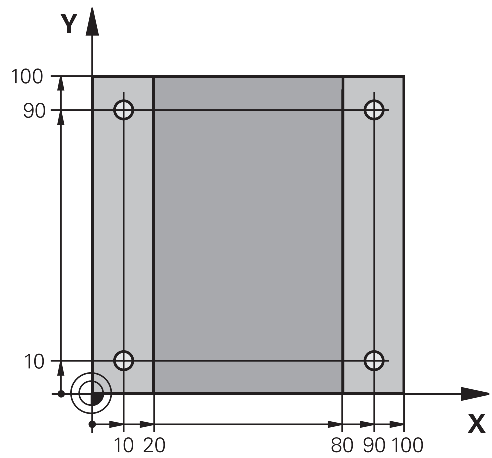
The following table shows examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (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, so the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (**Q200**=2, 20>2, so the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (**Q200**=2, 80>2, so the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (**Q200**=5, 8>5, so the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (**Q200**=5, 20>5, so the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (**Q200**=5, 80>5, so the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (**Q200**=20, 80>20, so the value 20 is used.)	-80

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 L 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 L X+10 R0 FMAX M3	; Approach hole 1, spindle ON
7 L Y+10 R0 FMAX M99	; Approach hole 1, cycle call
8 L X+90 R0 FMAX M99	; Approach hole 2, cycle call
9 L Y+90 R0 FMAX M99	; Approach hole 3, cycle call
10 L X+10 R0 FMAX M99	; Approach hole 4, cycle call
11 L Z+250 R0 FMAX M2	; Retract the tool, end program
12 END PGM C200 MM	

Example: Using cycles in conjunction with PATTERN DEF

The drill hole coordinates are stored in the PATTERN DEF POS pattern definition. The control calls the drill hole coordinates 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)
- **GLOBAL DEF 125 POSITIONING**: 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.
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)

Further information: "Cycles:Drilling Cycles /Thread Cycles", Page 379

0 BEGIN PGM 1 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	; Tool call: centering tool (tool radius 4)
4 L Z+50 R0 FMAX	; Move tool to clearance height
5 PATTERN DEF ~	
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 ~	
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 ~	
Q342=+0 ;ROUGHING DIAMETER ~	
Q253=+750 ;F PRE-POSITIONING	
7 GLOBAL DEF 125 POSITIONING ~	
Q345=+1 ;SELECT POS. HEIGHT	
8 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
9 L Z+100 R0 FMAX	; Retract the tool

10 TOOL CALL 227 Z S5000	; Tool call: drill (radius 2.4)
11 L X+50 R0 F5000	; Move tool to clearance height
12 CYCL DEF 200 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 ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
13 CYCL CALL PAT F500 M3	; Cycle call in connection with the point pattern
14 L Z+100 R0 FMAX	; Retract the tool
15 TOOL CALL 263 Z S200	; Tool call: tap (radius 3)
16 L Z+100 R0 FMAX	; Move tool to clearance height
17 CYCL DEF 206 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	
18 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
19 L Z+100 R0 FMAX	; Retract the tool
20 M30	; End of program
21 END PGM 1 MM	

13.11 Cycle 206 TAPPING

Application

The thread is cut in one or more passes. A floating tap holder is used.

Related topics

- Cycle **207 RIGID TAPPING** without floating tap holder

  Further information: "Cycle 207 RIGID TAPPING ", Page 429

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 set-up clearance at the end of the dwell time. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
4 At the set-up clearance, the direction of spindle rotation reverses once again.

> A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

Notes

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! There is a danger of collision!

- 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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.
- In Cycle **206**, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

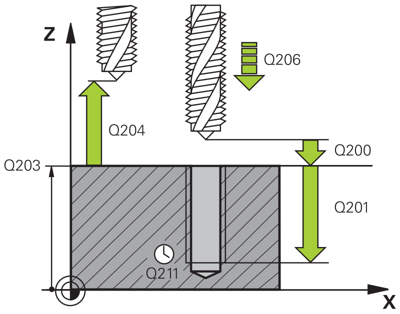
Notes on programming

- 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.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603):
    FeedPotentiometer (**default**) (speed override is not active), the control then adjusts the speed as required
    SpindlePotentiometer (feed rate override is not active)
  - **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.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Guide value: 4 times the thread pitch Input: **0...99999.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool during tapping Input: **0...99999.999** or **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: **0...3600.0000**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**

Example

```
11 CYCL DEF 206 TAPPING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q201=-18             ;DEPTH OF THREAD ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q211=+0              ;DWELL TIME AT DEPTH ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

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 Cycle 207 RIGID TAPPING

Application

Refer to your machine manual.

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

This cycle is effective only for machines with servo-controlled spindle.

The control cuts the thread without a floating tap holder in one or more passes.

Related topics

- Cycle **206 TAPPING** with floating tap holder

  Further information: "Cycle 206 TAPPING ", Page 426

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 It then reverses the direction of spindle rotation and the tool is retracted to set-up clearance. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
4 The control stops the spindle turning at that set-up clearance

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.

Notes

Cycle **207 RIGID TAPPING** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

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! There is a danger of collision!

- 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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- 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 will stand still after the end of the cycle. In this case, 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.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

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.

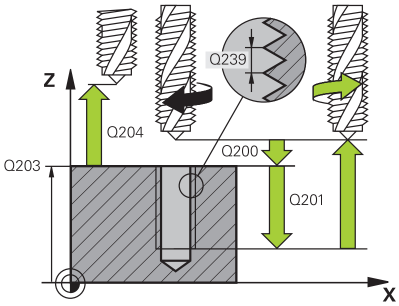
Notes on programming

- 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.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (spindle speed override is not active); the control then adjusts 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

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+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: **-99.9999...+99.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**

Example

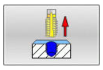
```
11 CYCL DEF 207 RIGID TAPPING ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q201=-18            ;DEPTH OF THREAD ~
    Q239=+1             ;THREAD PITCH ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

Retracting after a program interruption

Retracting in the Positioning with Manual Data Input operating mode

Proceed as follows:

- To interrupt thread cutting, press the **NC stop** key



- Press the retract soft key.

- Press **NC Start**
- > The tool retracts from the hole and moves 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

Proceed as follows:

- To interrupt the program, press the **NC stop** key

- Press the **MANUAL TRAVERSE** soft key
- Retract the tool in the active spindle axis

- To continue program execution, press the **RESTORE POSITION** soft key

- Then press **NC Start**
- > The control returns the tool to the position it had assumed before the **NC stop** key was pressed.

NOTICE

Danger of collision!

If you retract the tool manually and move it in the negative direction instead of the positive direction, there is a danger of collision.

- With a manual retraction you can move the tool in the positive as well as the negative direction of the tool axis.
- Before starting the manual retraction, you should make yourself fully aware of the direction into which you move the tool out of the hole.

13.13 Programming examples

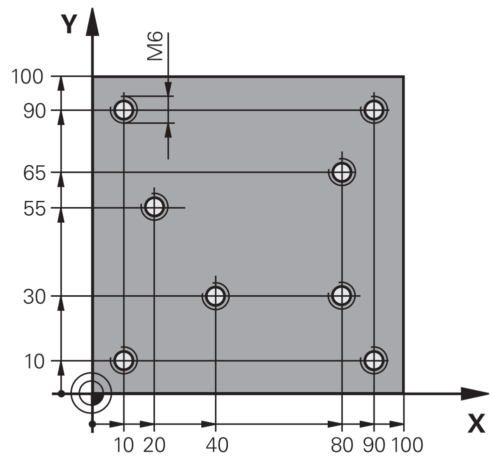
Example: Thread milling

The drill hole coordinates are stored in LBL 1 and are called by the control with **CALL LBL**.

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 TAP 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 171 Z S5000	; Tool call: centering tool
4 L Z+100 R0 FMAX M3	; Move tool to clearance height (program a value for F): the control positions the tool at the clearance height after every cycle
5 CYCL DEF 240 CENTERING ~	; Cycle definition: Centering
Q200=+2 ;SET-UP CLEARANCE ~	
Q343=+1 ;SELECT DIA./DEPTH ~	
Q201=-1 ;DEPTH ~	
Q344=-7 ;DIAMETER ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE	
6 CALL LBL 1	
7 L Z+100 R0 FMAX	; Retract the tool
8 TOOL CALL 227 Z S5000	; Tool call: drill
9 L Z+100 R0 FMAX M3	; Move tool to clearance height (enter a value for F)
10 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 ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	

Q395=+0 ;DEPTH REFERENCE	
11 CALL LBL 1	
12 L Z+100 R0 FMAX	; Retract the tool
13 TOOL CALL 263 Z S200	; Tool call: tap
14 L Z+100 R0 FMAX M3	; Move tool to clearance height
15 CYCL DEF 206 TAPPING ~	; Cycle definition: Tapping
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-22 ;DEPTH OF THREAD ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE	
16 CALL LBL 1	
17 L Z+100 R0 FMAX	; Retract the tool, end program
18 M30	
19 LBL 1	
20 L X+10 Y+10 R0 FMAX M99	
21 L X+40 Y+30 R0 FMAX M99	
22 L X+80 Y+30 R0 FMAX M99	
23 L X+90 Y+10 R0 FMAX M99	
24 L X+80 Y+65 R0 FMAX M99	
25 L X+90 Y+90 R0 FMAX M99	
26 L X+10 Y+90 R0 FMAX M99	
27 L X+20 Y+55 R0 FMAX M99	
28 LBL 0	
29 END PGM TAP MM	

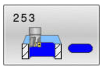
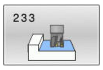
14

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
	Cycle 251 RECTANGULAR POCKET ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical	439
	Cycle 253 SLOT MILLING ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	444
	Cycle 256 RECTANGULAR STUD ■ Roughing and finishing cycle ■ Approach position: selectable	450
	Cycle 233 FACE MILLING ■ Roughing and finishing cycle ■ Roughing strategy and direction: selectable ■ Input of side walls	456

14.2 Cycle 251 RECTANGULAR POCKET

Application

Use Cycle **251** 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

Cycle sequence

Roughing

1 The tool plunges into 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 (**Q370**) and the finishing allowances (**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 set-up clearance above the current plunging depth. From there, the tool is returned 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.

Notes

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! There is a danger of collision!

- 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.

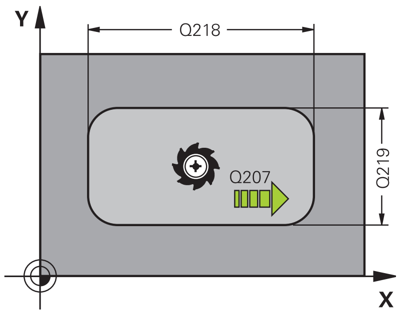
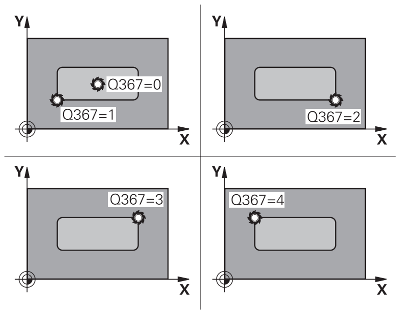
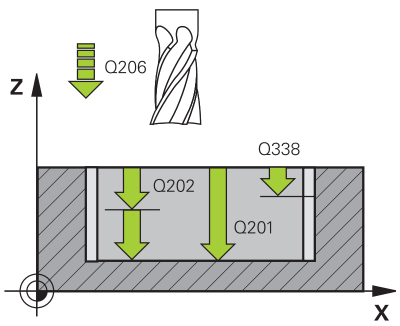
- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

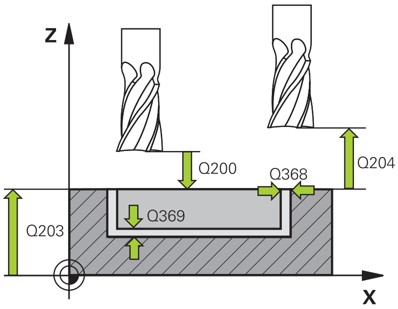
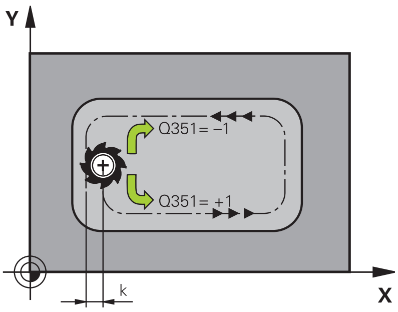
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q218 First side length?** Pocket length, parallel to the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q219 Second side length? Pocket length, parallel to the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between workpiece surface and bottom of pocket. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	**Q367 Position of pocket (0/1/2/3/4)?** Position of the pocket with respect to the tool when the cycle is called: **0**: Tool position = Center of pocket **1**: Tool position = Lower left corner **2**: Tool position = Lower right corner **3**: Tool position = Upper right corner **4**: Tool position = Upper left corner Input: **0**, **1**, **2**, **3**, **4**
	**Q202 Plunging depth?** Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min for moving to depth Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Help graphic	Parameter
	**Q368 Finishing allowance for side?** Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999**
	**Q351 Direction? Climb=+1, Up-cut=-1** Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1**
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.41**

Example

```
11 CYCL DEF 251 RECTANGULAR POCKET ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q218=+60                ;FIRST SIDE LENGTH ~
    Q219=+20                ;2ND SIDE LENGTH ~
    Q201=-20                ;DEPTH ~
    Q367=+0                 ;POCKET POSITION ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q207=+500               ;FEED RATE MILLING ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q338=+0                 ;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
12 L  X+50 R0 FMAX
13 L  Y+50 R0 FMAX M99
```

14.3 Cycle 253 SLOT MILLING

Application

Use Cycle **253** to completely machine a slot using straight-cut control. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, finishing
- Only roughing
- Only finishing

Cycle sequence

Roughing

1. The tool advances at the **FEED RATE FOR PLNGNG Q206** to the first plunging depth **Q202**. The slot created by the roughing process is exactly as wide as the diameter of the tool. When roughing, the control moves the tool only 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 slot is roughed out, taking parameters **Q351** and **Q352** into account.
3. Depending on the setting of parameter **Q352**, 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 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 set-up clearance **Q200**, moves it back to the center of the slot and then to 2nd set-up clearance **Q204**.

Finishing

6. If a finishing allowance has been defined during pre-machining, the control first finishes the slot walls, using 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.

Notes

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! There is a danger of collision!

- 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! There is a danger of collision!

- 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

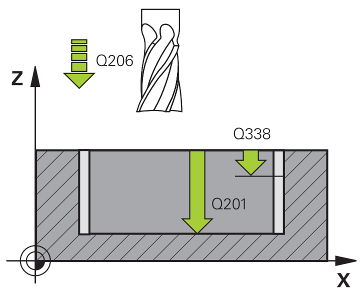
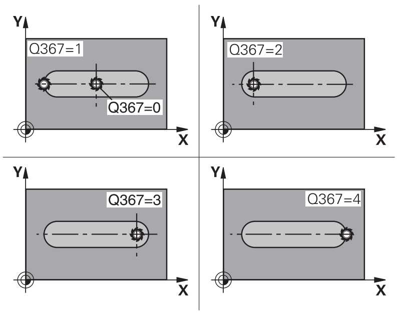
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- Please note that, after roughing, the slot is as wide as the tool diameter, regardless of parameter **Q219**.

Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Specify the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Input: **0**, **1**, **2**
	Q218 Length of slot? Enter the length of the slot. It is parallel to the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q219 Width of slot? Enter the width of the slot, which must be parallel to the secondary axis of the working plane. 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. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
 	**Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q374 Slot direction? Enter whether the slot is rotated by 90° (vertical slot, input: 1) or whether it is not rotated (horizontal slot, input: 0). The center of rotation is at the center of the slot. Input: **0**, **1**
	Q367 Position of slot (0/1/2/3/4)? Position of the figure relative to the position of the tool when the cycle is called: **0**: Tool position = Center of figure **1**: Tool position = Left end of figure **2**: Tool position = Center of left figure arc **3**: Tool position = Center of right figure arc **4**: Tool position = Right end of figure Input: **0**, **1**, **2**, **3**, **4**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min for moving to depth Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance?

Help graphic	Parameter
	Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1**
	Q352 Plunge position? Specify at which position along the main axis the tool should plunge: **+1**: Plunging position always at right end of slot **-1**: Plunging position always at left end of slot **0**: Reciprocating plunge Input: **-1**, **0**, **+1**

Example

```
11 CYCL DEF 253 SLOT MILLING ~
    Q215=+0             ;MACHINING OPERATION ~
    Q218=+60            ;SLOT LENGTH ~
    Q219=+10            ;SLOT WIDTH ~
    Q201=-20            ;DEPTH ~
    Q374=+0             ;SLOT DIRECTION ~
    Q367=+0             ;SLOT POSITION ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q207=+500           ;FEED RATE MILLING ~
    Q206=+150           ;FEED RATE FOR PLNGNG ~
    Q385=+500           ;FINISHING FEED RATE ~
    Q338=+0             ;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
12 L  X+50 R0 FMAX
13 L  Y+50 R0 FMAX M99
```

14.4 Cycle 256 RECTANGULAR STUD

Application

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.

Cycle sequence

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 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

Notes

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! There is a danger of collision! ▶ 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

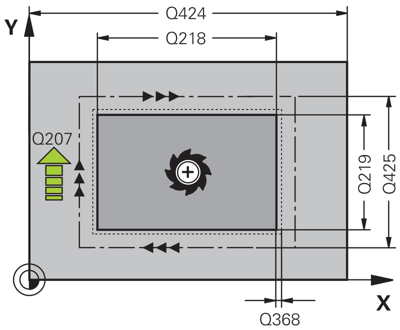
NOTICE
Danger of collision! If there is not enough room for the approach movement next to the stud, there is danger of collision. ▶ Depending on the approach position **Q439**, leave enough room next to the stud for the approach movement ▶ Leave room next to the stud for the approach motion ▶ At least tool diameter + 2 mm ▶ At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position.

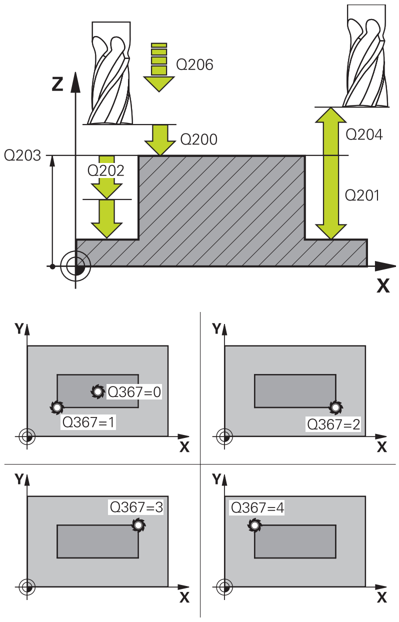
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

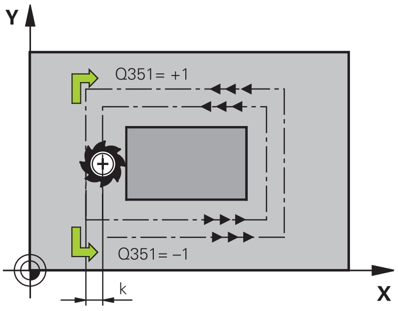
Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	**Q218 First side length?** Length of stud parallel to the main axis of the working plane This value has an incremental effect. Input: **0...99999.9999**
	Q424 Workpiece blank side length 1? Length of stud blank parallel to the main 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. This value has an incremental effect. Input: **0...99999.9999**
	Q219 Second side length? Length of stud parallel to the secondary 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. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	**Q425 Workpiece blank side length 2?** Length of stud blank parallel to the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q201 Depth? Distance between workpiece surface and bottom of stud. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q367 Position of stud (0/1/2/3/4)? Position of the stud with respect to the tool when the cycle is called. **0**: Tool position = Center of stud **1**: Tool position = Lower left corner **2**: Tool position = Lower right corner **3**: Tool position = Upper right corner **4**: Tool position = Upper left corner Input: **0**, **1**, **2**, **3**, **4**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while moving to depth Input: **0...99999.999** or **FAUTO**, **FMAX**, **FU**, **FZ**
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	**Q351 Direction? Climb=+1, Up-cut=-1** Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1**
	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: **0.1...1999**

Example

```
11 CYCL DEF 256 RECTANGULAR STUD ~
    Q215=+1                ;MACHINING OPERATION ~
    Q218=+60               ;FIRST SIDE LENGTH ~
    Q424=+75               ;WORKPC. BLANK SIDE 1 ~
    Q219=+20               ;2ND SIDE LENGTH ~
    Q425=+60               ;WORKPC. BLANK SIDE 2 ~
    Q201=-20               ;DEPTH ~
    Q367=+0                ;STUD POSITION ~
    Q202=+5                ;PLUNGING DEPTH ~
    Q207=+500              ;FEED RATE MILLING ~
    Q206=+3000             ;FEED RATE FOR PLNGNG ~
    Q385=+500              ;FINISHING FEED RATE ~
    Q368=+0                ;ALLOWANCE FOR SIDE ~
    Q369=+0                ;ALLOWANCE FOR FLOOR ~
    Q338=+0                ;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
12 L  X+50 R0 FMAX
13 L  Y+50 R0 FMAX M99
```

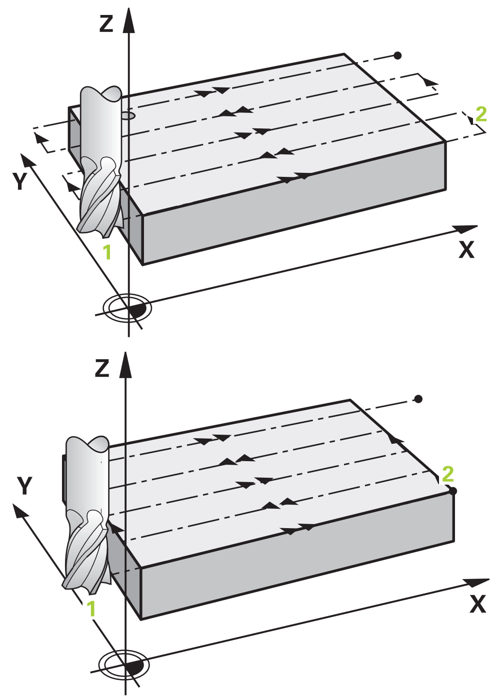
14.5 Cycle 233 FACE MILLING

Application

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

Strategies Q389=0 and Q389 =1



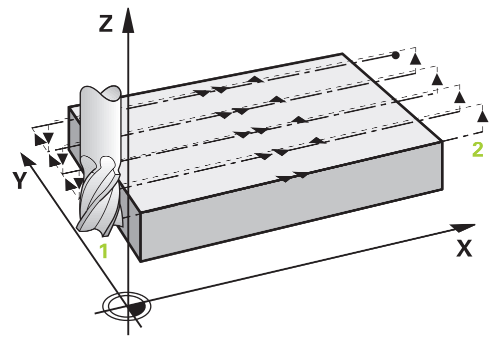
The strategies **Q389**=0 and **Q389**=1 differ in the overtravel during face milling. If **Q389**=0, the end point lies outside of the surface, with **Q389**=1, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389**=0 is used, the control additionally moves the tool beyond the level surface by the tool radius.

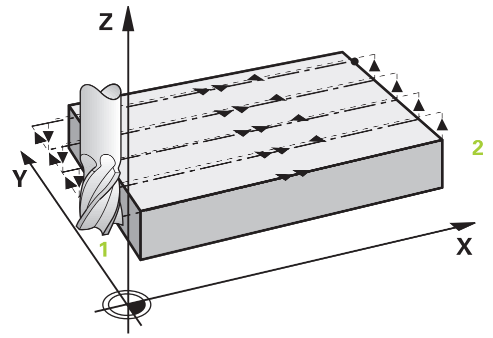
Cycle sequence

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 set-up clearance in the spindle axis.
3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
4 The control moves the tool to end point **2** at the programmed feed rate for milling.
5 The control then shifts the tool laterally to the starting point of the next line at the pre-positioning feed rate. 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 in the opposite direction at the feed rate for milling.
7 The process is repeated until the programmed surface has been machined completely.
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 spindle 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.



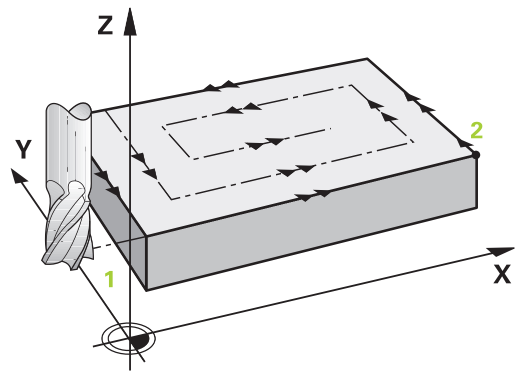


Cycle sequence

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 set-up clearance in the spindle axis.
3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
4 The tool subsequently advances at the programmed feed rate for milling **Q207** to the end point **2**.
5 The control positions the tool in the tool 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 **Q370** and the set-up clearance to the side **Q357**.
6 The tool then returns to the current infeed depth and moves in the direction of the 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 spindle 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

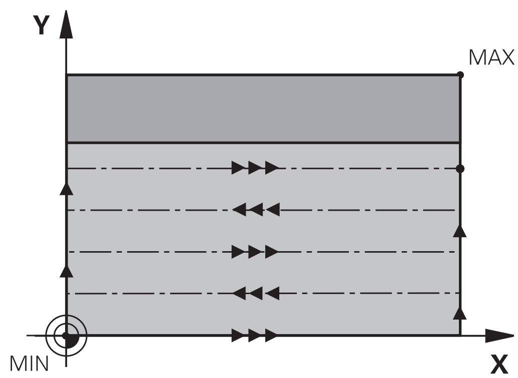
Cycle sequence



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 set-up clearance in the spindle axis.
3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
4 The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a linear tangential approach path.
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 completed. 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 spindle 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.



Notes

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! There is a danger of collision!

- 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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- 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.
- Cycle **233** monitors the entries made for the tool or cutting edge length in **LCUTS** in 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.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.

Notes on programming

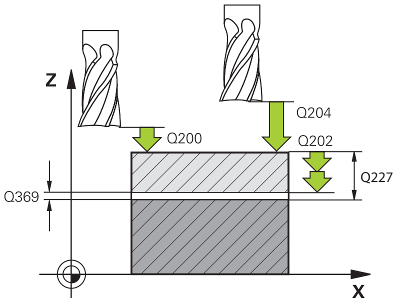
- Pre-position the tool in the working plane to the starting position with radius compensation R0. Note the machining direction.
- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- If you define **Q370 TOOL PATH OVERLAP** >1, the programmed overlap factor will be taken into account right from the first machining path.
- If a limit (**Q347**, **Q348** or **Q349**) was programmed in the machining direction **Q350**, the cycle will extend the contour in the infeed direction by corner radius **Q220**. The specified surface will be machined completely.

Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q389 Machining strategy (0-4)? Specify how the control machines the surface: **0**: Meander machining, stepover at positioning feed rate outside the surface to be machined **1**: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined **2**: Machining line by line, retraction and stepover at positioning feed rate outside the surface to be machined **3**: Machining line by line, retraction and stepover at positioning feed rate at the edge of the surface to be machined **4**: Helical machining, uniform infeed from the outside toward the inside Input: **0**, **1**, **2**, **3**, **4**
	Q350 Milling direction? Axis in the working plane that defines the machining direction: **1**: Main axis = Machining direction **2**: Secondary axis = Machining direction Input: **1**, **2**
	Q218 First side length? Length of the surface to be machined in the main axis of the working plane, referencing the starting point in the 1st axis. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q219 Second side length? Length of the surface to be machined in the secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referenced to the **STARTNG PNT 2ND AXIS**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter



Q227 Starting point in 3rd axis?

Coordinate of the workpiece surface used to calculate the infeeds. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q386 End point in 3rd axis?

Coordinate in the spindle axis on which the surface will be face-milled. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q369 Finishing allowance for floor?

Finishing allowance in depth which remains after roughing.

This value has an incremental effect.

Input: **0...99999.9999**

Q202 Maximum plunging depth?

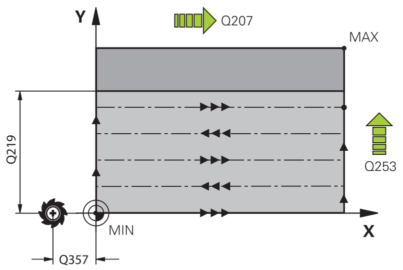
Infeed per cut. Enter an incremental value greater than 0.

Input: **0...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: **0.0001...1.9999**



Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Q385 Finishing feed rate?

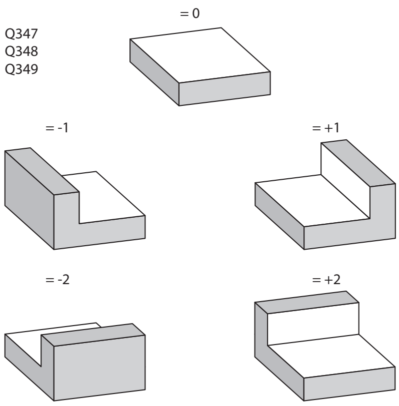
Traversing speed of the tool in mm/min while milling the last infeed

Input: **0...99999.999** or **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: **0...99999.9999** or **FMAX**, **FAUTO**

Help graphic	Parameter
	Q357 Safety clearance to the side? Parameter **Q357** influences the following situations: **Approaching the first infeed depth: Q357** is the lateral distance from the tool to the workpiece. **Roughing with the Q389 = 0 to 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 are extended by **Q357** in the **Q350 MILLING DIRECTION**. This value has an incremental effect. Input: **0...99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999**
	**Q347 1st limit?** Select the side of the workpiece where the plane surface is bordered by a side wall. Depending on the position of the side wall, the control limits the machining of the plane surface to the corresponding starting point coordinate or side length: **0**: No limitation **-1**: Limit in negative main axis **+1**: Limit in positive main axis **-2**: Limit in negative secondary axis **+2**: Limit in positive secondary axis Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q348 2nd limit? See parameter **Q347** 1st limit Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q349 3rd limit? See parameter **Q347** 1st limit Input: **-2**, **-1**, **0**, **+1**, **+2**

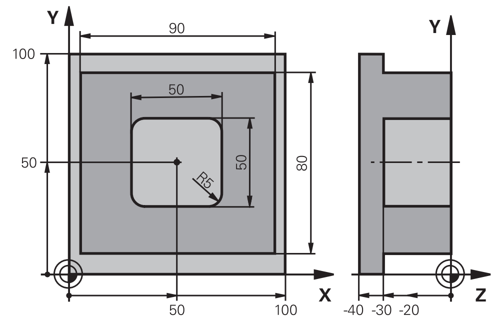
Help graphic	Parameter
	Q368 Finishing allowance for side? Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q367 Surface position (-1/0/1/2/3/4)? Position of the surface relative to the position of the tool when the cycle is called: **-1**: Tool position = Current position **0**: Tool position = Center of stud **1**: Tool position = Lower left corner **2**: Tool position = Lower right corner **3**: Tool position = Upper right corner **4**: Tool position = Upper left corner Input: **-1**, **0**, **+1**, **+2**, **+3**, **+4**

Example

```
11 CYCL DEF 233 FACE MILLING ~
    Q215=+0             ;MACHINING OPERATION ~
    Q389=+2             ;MILLING STRATEGY ~
    Q350=+1             ;MILLING DIRECTION ~
    Q218=+60            ;FIRST SIDE LENGTH ~
    Q219=+20            ;2ND SIDE LENGTH ~
    Q227=+0             ;STARTNG PNT 3RD AXIS ~
    Q386=+0             ;END POINT 3RD AXIS ~
    Q369=+0             ;ALLOWANCE FOR FLOOR ~
    Q202=+5             ;MAX. PLUNGING DEPTH ~
    Q370=+1             ;TOOL PATH OVERLAP ~
    Q207=+500           ;FEED RATE MILLING ~
    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
12 L  X+50 R0 FMAX
13 L  Y+50 R0 FMAX M99
```

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	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call: roughing/finishing
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD	Cycle definition: outside machining
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 MILLING	
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	Cycle call for outside machining
8 CYCL DEF 252 RECTANGULAR POCKET	Cycle definition: rectangular pocket
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 MILLING	
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	

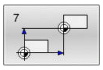
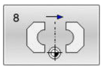
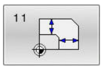
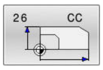
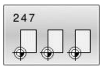
15

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
	Cycle 7 DATUM SHIFT ■ Shifting contours directly in the NC program ■ Or shifting contours using datum tables	471
	Cycle 8 MIRRORING ■ Mirroring contours	476
	Cycle 11 SCALING FACTOR ■ Resizing contours	477
	Cycle 26 AXIS-SPECIFIC SCALING ■ Axis-specific resizing of contours	478
	Cycle 247 PRESETTING ■ Datum setting during program run	474

Effectiveness of coordinate transformations

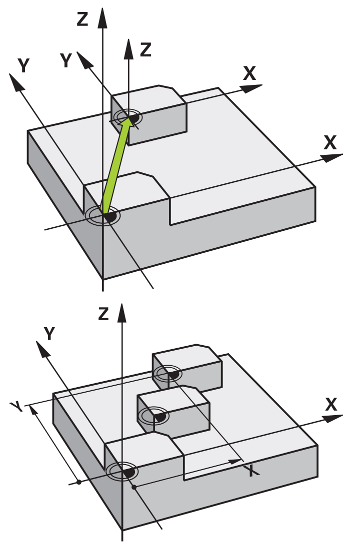
Beginning of effect: A coordinate transformation takes effect as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

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 Cycle 7 DATUM SHIFT

Application



Refer to your machine manual.

A datum shift allows machining operations to be repeated at various locations on the workpiece. Within an NC program, you can either program datum points directly in the cycle definition or call them from a datum table.

Use datum tables for the following purposes:

- Frequent use of the same datum shift
- Frequently recurring machining sequences on different workpieces
- Frequently recurring machining sequences at various locations on one 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.

Reset

- 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.

Status display

The additional status display **TRANS** contains the following information:

- Coordinates from the datum shift
- Name and path of the active datum table
- Active datum number for datum tables
- Comment from the **DOC** column of the active datum number from the datum table

Related topics

- Datum shift with **TRANS DATUM**

  Further information: "Datum shift with TRANS DATUM", Page 310

Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- The main axis, secondary axis and tool axis are in effect in the W-CS or WPL-CS coordinate system. Rotary axes and parallel axes are in effect in the M-CS system.

Notes about machine parameters

- In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.

Additional information regarding datum shifts with datum tables:

- 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.
- 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 manager 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 manager in the **Program run, single block** and **Program run, full sequence** operating modes to select the desired table for program run: The table receives the status M
- The coordinate values from datum tables are only effective with absolute coordinate values.

Cycle parameters

Datum shift without a datum table

Help graphic	Parameter
	Shift? Enter the coordinates of the new datum. Absolute values are referenced to the workpiece datum, which is determined by the presetting. Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted). Up to six NC axes are possible. Input: **-999999999...+999999999**

Example

```
11 CYCL DEF 7.0 DATUM SHIFT
12 CYCL DEF 7.1  X+60
13 CYCL DEF 7.2  Y+40
14 CYCL DEF 7.3  Z+5
```

Datum shift with a datum table

Help graphic	Parameter
	Shift? Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the control activates the datum number entered in the Q parameter. Input: **0...9999**

Example

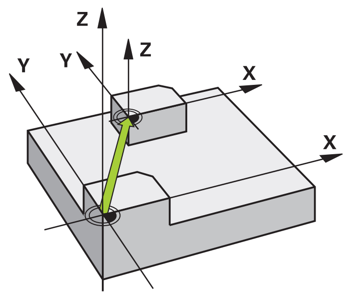
```
11 CYCL DEF 7.0 DATUM SHIFT
12 CYCL DEF 7.1 #5
```

15.3 Cycle 247 PRESETTING

Application

Use Cycle **247 PRESETTING** to activate a preset defined in the preset table as the new preset.

After cycle definition, all coordinate input and datum shifts (absolute or incremental) reference the new preset.



Status display

In the status display; the control shows the active preset number behind the preset symbol.

Related topics

- Activate the preset
  Further information: "Activating a preset", Page 319
- Copy the preset
  Further information: "Copying a preset", Page 321
- Correct the preset
  Further information: "Correcting a preset", Page 322
- Setting and activating presets
  Further information: User's Manual for **Setup, Testing and Running NC Programs**

Notes

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- ▶ Before activating a preset, check whether all columns contain values.
- ▶ For undefined columns, enter values (e.g., **0**)
- ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- 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 in effect in the Test Run operating mode.

Cycle parameters

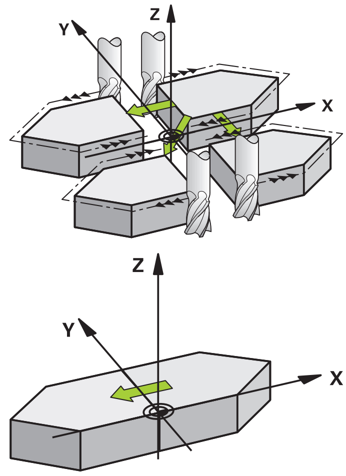
Help graphic	Parameter
	Number for preset? Enter the number of the desired preset from the preset table. Alternatively, you can use the **SELECT** soft key to directly select the desired preset from the preset table. Input: **0...65535**

Example

```
11 CYCL DEF 247 PRESETTING ~
    Q339=+4                ;PRESET NUMBER
```

15.4 Cycle 8 MIRRORING

Application



The control can machine the mirror image of a contour in the working plane.

Mirroring takes effect as soon as it has been defined in the NC program. It is also in effect 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.

Reset

Program Cycle **8 MIRRORING** again with **NO ENT**.

Related topics

- Mirroring with **TRANS MIRROR**

  Further information: "Mirroring with TRANS MIRROR", Page 313

Cycle parameters

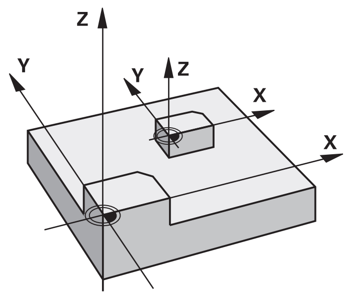
Help graphic	Parameter
	Mirror image axis? Enter the axes to be mirrored. You can mirror all axes—including rotary axes—with the exception of the spindle axis and its associated secondary axis. You can enter up to three NC axes. Input: **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example

```
11 CYCL DEF 8.0 MIRRORING
12 CYCL DEF 8.1 X Y Z
```

15.5 Cycle 11 SCALING FACTOR

Application



The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The scaling factor takes effect as soon as it has been defined in the NC program. It is also in effect 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

Requirement

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)

> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Reset

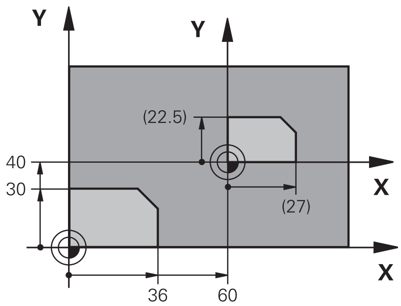
Program Cycle **11 SCALING FACTOR** again and specify a scaling factor of 1.

Related topics

- Scaling with **TRANS SCALE**

  Further information: "Scaling with TRANS SCALE", Page 315

Cycle parameters

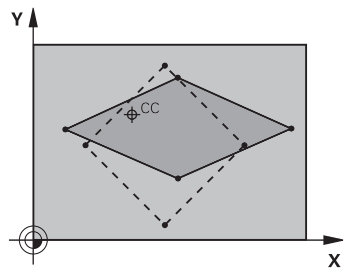
Help graphic	Parameter
	**Factor?** Enter the scaling factor SCL. The control multiplies the coordinates and radii with SCL. Input: **0.000001...99.999999**

Example

```
11 CYCL DEF 11.0 SCALING FACTOR
12 CYCL DEF 11.1 SCL 0.75
```

15.6 Cycle 26 AXIS-SPECIFIC SCALING

Application



Use Cycle **26** to account for shrinkage and allowance factors for each axis.

The scaling factor takes effect as soon as it has been defined in the NC program. It is also in effect in the **Positioning w/ Manual Data Input** operating mode. The active scaling factor is shown in the additional status display.

Reset

Program Cycle **11 SCALING FACTOR** again and enter a scaling factor of 1 for the corresponding axis.

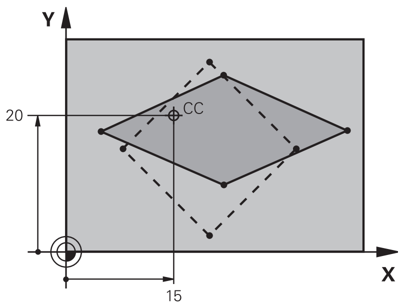
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The contour is enlarged or reduced relative to the center, and not necessarily (as in Cycle **11 SCALING FACTOR**) relative to the active datum.

Notes on 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.

Cycle parameters

Help graphic	Parameter
	**Axis and factor?** Select the coordinate axis/axes via soft key. Enter the factor(s) for axis-specific enlargement or reduction. Input: **0.000001...99.999999**
	Centerpoint coord. of extension? Center of the axis-specific enlargement or reduction. Input: **-999999999...+999999999**

Example

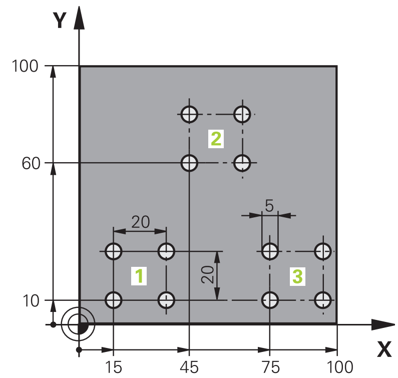
```
11 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
12 CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```

15.7 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	

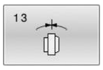
16

Cycles: Special Functions

16.1 Fundamentals

Overview

The control provides the following cycles for the following special purposes:

Soft key	Cycle	Page
	Cycle 9 DWELL TIME ■ Delay execution by the programmed dwell time	483
	Cycle 12 PGM CALL ■ Call any NC program	484
	Cycle 13 ORIENTATION ■ Rotate spindle to a specific angle	486

16.2 Cycle 9 DWELL TIME

Application

> This cycle can be executed in the **FUNCTION MODE MILL** machining mode.

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 takes effect as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Related topics

- Dwell time with **FUNCTION FEED DWELL**

  Further information: "Dwell time FUNCTION FEED DWELL", Page 305
- Dwell time with **FUNCTION DWELL**

  Further information: "Dwell time FUNCTION DWELL", Page 340

Cycle parameters

Help graphic	Parameter
	Dwell time in secs.? Enter the dwell time in seconds. Input: **0...3600 s** (**1** hour) in steps of 0.001 seconds

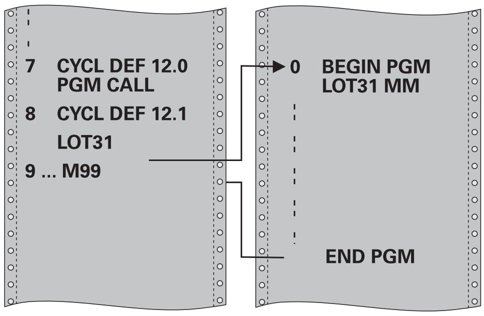
Example

```
89 CYCL DEF 9.0  DWELL TIME
90 CYCL DEF 9.1  DWELL 1.5
```

16.3 Cycle 12 PGM CALL

Application

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.



Related topics

- Calling external NC programs

  Further information: "Calling an external NC program", Page 183

Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- 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.

Notes on 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**.

Cycle parameters

Help graphic	Parameter
	Program name Enter the name of the NC program to be called and, if necessary, the path where it is located, Use the Select soft key to activate the File Select dialog. Select the NC program to be called. The **SYNTAX** soft key allows you to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path. If the complete path is within the quotation marks, you can use both **** and **/** to separate the folders and files.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

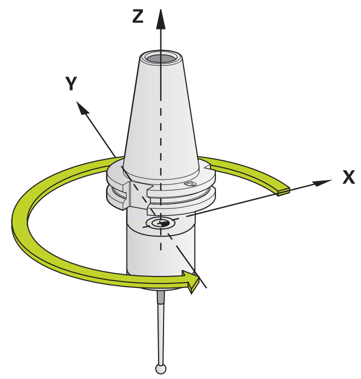
Declare NC program Stempel_stamp.h as a cycle and call it with M99

```
11 CYCL DEF 12.0 PGM CALL
12 CYCL DEF 12.1 PGM TNC:\nc_prog\demo\Stempel_stamp.h
13 L  X+20 FMAX
14 L  Y+50 FMAX M99
```

16.4 Cycle 13 ORIENTATION

Application

⚙	Refer to your machine manual. Machine and control must be specially prepared by the machine manufacturer 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 purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the transceiver window of HEIDENHAIN 3D 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 manufacturer.

Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter
	Orientation angle Enter the angle relative to the angle reference axis of the working plane. Input: **0...360**

Example

```
11 CYCL DEF 13.0 ORIENTATION
12 CYCL DEF 13.1 ANGLE180
```

17

Touch Probe Cycles

17.1 General information about touch probe cycles

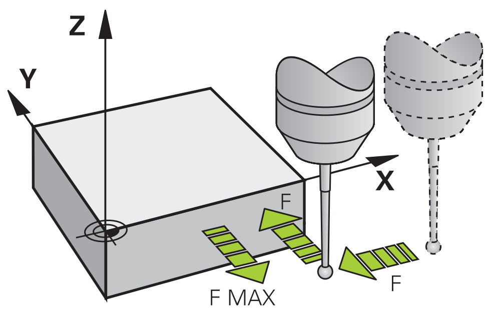
The control must be specifically prepared by the machine manufacturer for the use of a 3D touch probe.

If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (option 17) is automatically enabled.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

Method of function

- Refer to your machine manual.
- The control must be specifically prepared by the machine manufacturer for the use of a 3D touch probe.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- The touch probe cycles are available only with option 17. If you are using a HEIDENHAIN touch probe, this option is automatically available.
- The control's full range of functions is available only if the **Z** tool axis is used.
- Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.



Whenever the control runs a touch probe cycle, the 3D touch probe approaches the workpiece parallel to the axis. The machine manufacturer will determine the probing feed rate in a machine parameter.

Further information: "Before you start working with touch probe cycles", Page 489

When the probe stylus contacts the workpiece,

- the 3D 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** operating modes, the control provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Set presets

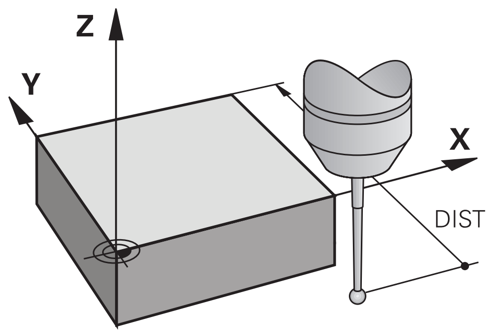
17.2 Before you start working with touch probe cycles

To make it possible to cover the widest possible range of measuring tasks, you have various possibilities for defining the behavior common to all touch probe cycles.

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

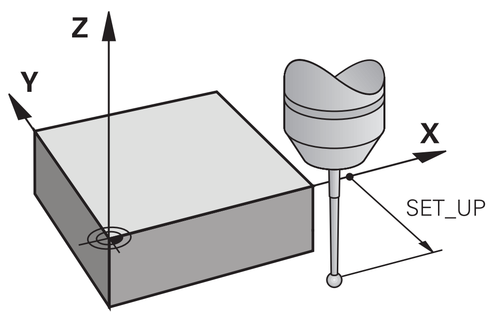
Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the range defined in **DIST**, 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 be greater than defined in the optional machine parameter **maxTouchFeed** (no. 122602).

The feed-rate potentiometer can be effective in touch probe cycles. The machine manufacturer enters the necessary settings. (Parameter **overrideForMeasure** (no. 122604), must be configured correspondingly.)

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. The control runs the cycle automatically as soon as it reads the cycle definition in the program run.

Notes

NOTICE
Danger of collision! When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision! ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**. ▶ Reset any coordinate transformations beforehand.

Notes in connection with programming and execution

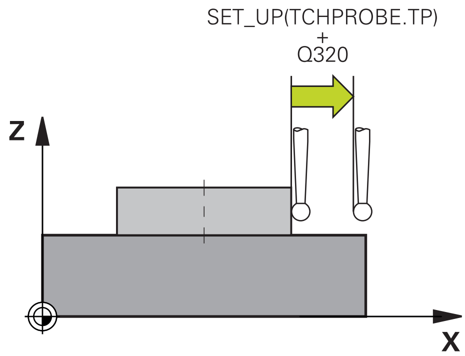
Pre-positioning

Before each probing operation, the control pre-positions the touch probe.

Pre-positioning is done in the inverse probing direction.

The distance between the probing point and the pre-position results from the following values:

- Ball-tip radius **R**
- **SET_UP** from the touch-probe table
- **Q320 SET-UP CLEARANCE**



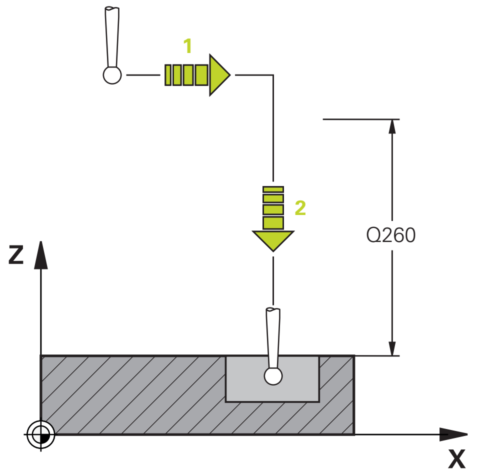
Positioning logic

Touch-probe cycles with numbers from **400** through **499** pre-position the touch probe according to the following positioning logic:

Current position > Q260 CLEARANCE HEIGHT

1 The control positions the touch probe at **FMAX** to the pre-position in the working plane.

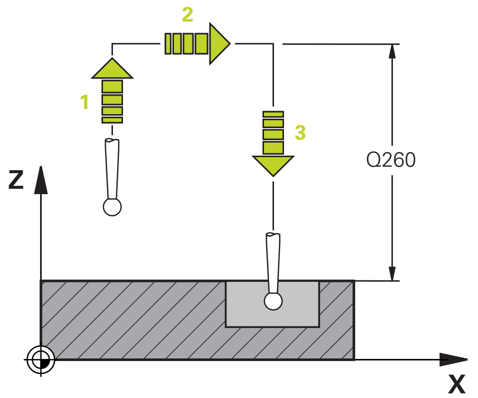
   Further information: "Pre-positioning ", Page 491
2 Then, the control positions the touch probe at **FMAX** in the tool axis, directly to the probing height.



Current position < Q260 CLEARANCE HEIGHT

1 The control positions the touch probe at **FMAX** to **Q260 CLEARANCE HEIGHT**.
2 The control positions the touch probe at **FMAX** to the pre-position in the working plane.

   Further information: "Pre-positioning ", Page 491
3 Then, the control positions the touch probe at **FMAX** in the tool axis, directly to the probing height.



17.3 Fundamentals

Overview

Refer to your machine manual.

Some cycles and functions may not be provided on your machine.

Option 17 is required.

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

Operating notes

- When running touch probe cycles, Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active
- HEIDENHAIN only guarantees the proper operation of the probing cycles if HEIDENHAIN touch probes are used.

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 are stored in the tool table 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 the **Programming** operating mode using the **CYCL DEF** key. The following cycles are available:

Soft key	Cycle	Page
	Cycle 480 CALIBRATE TT (option 17) ■ Calibrating the tool touch probe	500
	Cycle 481 CAL. TOOL LENGTH (option 17) ■ Measuring the tool length	505
	Cycle 482 CAL. TOOL RADIUS (option 17) ■ Measuring the tool radius	508
	Cycle 483 MEASURE TOOL (option 17) ■ Measuring the tool length and radius	512
	Cycle 484 CALIBRATE IR TT (option 17) ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	502

> Operating notes:
> - The touch probe cycles can be used only when the central tool file TOOL.T is active.
> - Before working with the touch probe cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.

Measuring a tool of length 0

Refer to your machine manual!

The optional machine parameter **maxToolLengthTT** (no. 122607) enables the machine manufacturer to define a maximum tool length for the tool measurement cycles.

HEIDENHAIN recommends that you always define tools with their actual tool length if possible.

The tool measuring cycles measure tools automatically. You can also measure tools defined with a length **L** of 0 in the tool table. To do this, the machine manufacturer must define a maximum tool length value in the optional machine parameter **maxToolLengthTT** (no. 122607). The control starts a search in which the actual tool length is roughly determined in the first step. This is followed by a fine measurement.

Cycle run

1. The tool travels to a clearance height centered above the touch probe.

   The clearance height equals the value of the optional machine parameter **maxToolLengthTT** (no. 122607).
2. The control performs a rough measurement with the spindle standing still.

   When measuring a stationary tool, the control will use the feed rate for probing defined in the machine parameter **probingFeed** (no. 122709).
3. The control saves the roughly measured length.
4. The control performs a fine measurement with the values from the tool measuring cycle.

Notes

NOTICE

Risk of collision!

If the machine manufacturer fails to define the optional machine parameter **maxToolLengthTT** (no. 122607), there will be no tool search. The control pre-positions the tool with a length of 0. Risk of collision!

- Observe the machine parameter value in the machine manual.
- Define tools with the actual tool length **L**

NOTICE

Risk of collision!

Risk of collision if the tool is longer than the value of the optional machine parameter **maxToolLengthTT** (no. 122607)!

- Observe the machine parameter value in the machine manual

Setting machine parameters

- The touch probe cycles **480**, **481**, **482**, **483**, **484** can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).

Programming and operating notes:

- Before you start working with the touch probe cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300).
- When measuring a stationary tool, the control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

Setting of the spindle speed

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

Abbreviation	Definition
n	Shaft speed [rpm]
maxPeriphSpeed-Meas	Maximum permissible cutting speed in m/min
r	Active tool radius [mm]

Setting of the feed rate

The probing feed rate is calculated as follows:

v = measuring tolerance • n

Abbreviation	Definition
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. The control provides the following options:

- **ConstantTolerance**
- **VariableTolerance**
- **ConstantFeed**

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.

- **VariableTolerance**:

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**

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

Abbreviation	Definition
r	Active tool radius [mm]
measureTolerance1	Maximum permissible error of measurement

Setting for consideration of parallel axes and changes in the kinematics

Refer to your machine manual.

Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

Auxiliary or parallel axes cannot be probed, regardless of the setting of the optional machine parameter **calPosType** (no. 122606).

If the machine manufacturer changes the setting of the optional machine parameter, you need to recalibrate the tool touch probe.

Entries in the tool table for milling tools

Abbr.	Inputs	Dialog
CUT	The number of teeth of the tool for automatic tool measurement or cutting data calculation (maximum of 20 teeth)	**Number of teeth?**
LTOL	Permitted tool length deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...5.0000**	**Wear tolerance: length?**
RTOL	Permitted tool radius deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...5.0000**	**Wear tolerance: radius?**
DIRECT.	Cutting direction of the tool for automatic tool measurement with a rotating tool. Input: **-**, **+**	**Cutting direction (M3 = -)?**
R-OFFS	Position of tool upon length measurement, offset between the probe contact center and the tool center for automatic tool measurement. Default setting: No value entered (offset = tool radius) Input: **-99999.9999...+99999.9999**	**Tool offset: radius?**
L-OFFS	Position of tool upon radius measurement, distance between the probe contact top edge and the tool tip for automatic tool measurement. Is added to the **offsetToolAxis** machine parameter (no. 122707). Input: **-99999.9999...+99999.9999**	**Tool offset: length?**
LBREAK	Permitted tool length deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...9.0000**	**Breakage tolerance: length?**
RBREAK	Permitted tool radius deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...9.0000**	**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: four cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from **offsetToolAxis** (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: four cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

17.4 Cycle 480 CALIBRATE TT (option 17)

Application

Refer to your machine manual!

You calibrate the TT with touch probe cycle **480**. The calibration process runs automatically. The control also measures the center offset of the calibration tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

You calibrate the TT with touch probe cycle **480**.

Touch probe

For the touch probe you use a spherical probe contact

Calibration tool

The calibration 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.

Cycle run

1. Clamp the calibration tool. The calibration tool must be a precisely cylindrical part, for example a cylindrical pin
2. Manually position the calibration tool in the working plane over the center of the TT
3. Position the calibration tool in the tool axis at approximately 15 mm plus set-up clearance over 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. This is followed by calibration in the working plane
7. The control positions the calibration tool in the working plane at a position of TT radius + set-up clearance + 11 mm
8. Then the control 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

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.

Notes about machine parameters

- Use the machine parameter **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300) to define the functionality of the calibration cycle. Refer to your machine manual.
  - Use the machine parameter **centerPos** to define the position of the TT within the machine's working space.
- The TT needs to be recalibrated if you change the position of the TT on the table and/or a **centerPos** machine parameter.
- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the calibration tool above the top of the probe contact (safety zone from **safetyDistToolAx** (no. 114203)). Input: **-99999.9999...+99999.9999**

ExampleNew format

```
11 TOOL CALL 12 Z
12 TCH PROBE 480 CALIBRATE TT ~
    Q260=+100              ;CLEARANCE HEIGHT
```

17.5 Cycle 484 CALIBRATE IR TT (option 17)

Application

Cycle **484** allows you to calibrate your tool touch probe (e.g., the wireless infrared TT 460 tool touch probe). You can perform the calibration process with or without manual intervention.

- **With manual intervention**: If you define **Q536** = 0, then the control will stop before the calibration process. You then need to position the calibration tool manually above the center of the tool touch probe.
- **Without manual intervention**: If you define **Q536** = 1, then the control will automatically execute the cycle. You may have to program a prepositioning movement before. This depends on the value of the parameter **Q523 POSITION TT**.

Cycle run

> Refer to your machine manual.
> The machine manufacturer defines the functionality of the cycle.

To calibrate the tool touch probe, program the touch probe cycle **484**. In input parameter **Q536**, you can specify whether you want to run the cycle with or without manual intervention.

Touch probe

For the touch probe you use a spherical probe contact

Calibration tool:

The calibration tool must be a precisely cylindrical part, for example a cylindrical pin. Enter the exact length and radius of the calibration 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 calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Q536 = 0: With manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibration tool
- ▶ Start the calibration cycle
- > The control interrupts the calibration cycle and displays a dialog in a new window.
- ▶ Manually position the calibration tool above the center of the tool touch probe.

> Ensure that the calibration tool is located above the measuring surface of the probe contact.

- ▶ Press **NC Start** to resume cycle run
- > If you have programmed **Q523** = **2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

Q536 = 1: Without manual intervention before calibration

Proceed as follows:

- Insert the calibrating tool
- Position the calibration tool above the center of the tool touch probe before the start of the cycle.

> - Ensure that the calibration tool is located above the measuring surface of the probe contact.
> - For a calibration process without manual intervention, you do not need to position the calibration tool above the center of the tool touch probe. The cycle adopts the position from the machine parameters and automatically moves the tool to this position.

- Start the calibration cycle
- > The calibration cycle is executed without stopping.
- > If you have programmed **Q523** = **2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

Notes

NOTICE

Danger of collision!

If you program **Q536=1**, the tool must be pre-positioned before calling the cycle. 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. There is a danger of collision!

- Specify whether to stop before cycle start or run the cycle automatically without stopping.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. If you use a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. Major inaccuracies may occur if you use a calibration tool whose diameter is too small and/or that protrudes too far from the chuck.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.
- The TT needs to be recalibrated if you change its position on the table.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	Q536 Stop before running (0=Stop)? Define whether the control will stop before the calibration process or whether the cycle will automatically be executed without a stop: **0**: Stop before the calibration process. The control prompts you to position the calibration tool manually 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**: Without stopping before the calibration process. The control starts the calibration process depending on **Q523**. Before running Cycle **484**, you may have to position the tool above the tool touch probe. Input: **0**, **1**
	Q523 Position of tool probe (0-2)? Position of the tool touch probe: **0**: Current position of the calibration tool. The tool touch probe is below the current position of the calibration tool. If **Q536 = 0**, position the calibration tool manually above the center of the tool touch probe during the cycle. If **Q536 = 1**, you need to position the calibration tool above the center of the tool touch probe before the start of the cycle. **1**: Configured position of the tool touch probe. The control adopts the position from the machine parameter **centerPos** (no. 114201). You do not need to pre-position the tool. The calibration tool approaches the position automatically. **2**: Current position of the calibration tool. See **Q523 = 0**. **0**. The control additionally writes the determined position (where applicable) to the machine parameter **centerPos** (no. 114201) after calibration. Input: **0**, **1**, **2**

17.6 Cycle 481 CAL. TOOL LENGTH (option 17)

Application

Refer to your machine manual!

For measuring the tool length, program touch probe cycle **482**. 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.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- Set **stopOnCheck** (no. 122717) to **TRUE**
- You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- 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 for tools with **up to 20 teeth**.
- Cycles **31** and **481** do not support touch probes, turning or dressing tools.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured 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 is also available in the Q 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. Input: **0**, **1**, **2**
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input: **-99999.9999...+99999.9999**
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: **0**, **1**

Example

```
11 TOOL CALL 12 Z
12 TCH PROBE 481 CAL. TOOL LENGTH ~
    Q340=+1              ;CHECK ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q341=+1              ;PROBING THE TEETH
```

17.7 Cycle 482 CAL. TOOL RADIUS (option 17)

Application

Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **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 face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). 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.

Further information: "Notes for individual tooth measurement Q341=1", Page 510

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

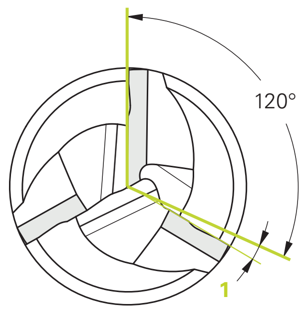
- Set **stopOnCheck** (no. 122717) to **TRUE**
- You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- 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.
- Cycles **32** and **482** do not support touch probes, turning or dressing tools.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement Q341=1



1 Angle deviation

NOTICE

Caution: Danger to the tool and workpiece!

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- Check the workpiece dimensions (for example, by using a workpiece touch probe)
- Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

$$\varepsilon = 90 \; - \; atan\left(\frac{h[tt]}{\frac{R \times 2 \times \pi}{x}}\right)$$

Abbreviation	Definition
ε	Maximum angle of twist
h[tt]	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool

On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

NOTICE

Caution: Possible material damage!

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.

- Check the workpiece dimensions during subsequent machining operations

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured 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 is also available in the Q 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 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. Input: **0**, **1**, **2**
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input: **-99999.9999...+99999.9999**
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: **0**, **1**

Example

```
11 TOOL CALL 12 Z
12 TCH PROBE 482 CAL. TOOL RADIUS ~
    Q340=+1            ;CHECK ~
    Q260=+100          ;CLEARANCE HEIGHT ~
    Q341=+1            ;PROBING THE TEETH
```

17.8 Cycle 483 MEASURE TOOL (option 17)

Application

Refer to your machine manual!

To measure the tool completely (length and radius), program touch probe cycle **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. Input parameters allow you to select which of the two following methods will be used to measure the tool:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

Measuring the tool while it is rotating:

The control measures the tool in a fixed programmed sequence. First, if possible, it measures the tool length, and then the tool radius.

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 touch probe cycles **481** and **482**.

Further information: "Notes for individual tooth measurement of radius Q341=1", Page 514

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

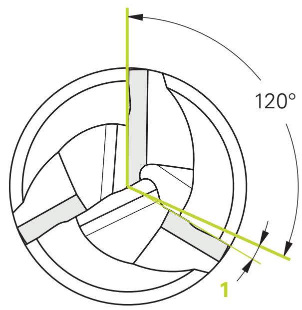
- Set **stopOnCheck** (no. 122717) to **TRUE**
- You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- 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.
- Cycles **33** and **483** do not support touch probes, turning or dressing tools.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement of radius Q341=1



1 Angle deviation

NOTICE

Caution: Danger to the tool and workpiece!

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- Check the workpiece dimensions (for example, by using a workpiece touch probe)
- Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

ε = 90 - **atan** (**h[tt]** / (tool radius * 2 * π / number of teeth)

Abbreviation	Definition
ε	Maximum angle of twist
h[tt]	Height of tool touch probe contact

On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

NOTICE

Caution: Possible material damage!

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.

- Check the workpiece dimensions during subsequent machining operations

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured 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 the Q parameters **Q115** and **Q116**. If the delta value is greater than the permissible tool length or tool 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 in TOOL.T. The control calculates the deviation from the stored values and writes it to the Q parameter **Q115** or **Q116**. Nothing is entered under L, R, or DL, DR in the tool table. Input: **0**, **1**, **2**
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input: **-99999.9999...+99999.9999**
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: **0**, **1**

Example

```
11 TOOL CALL 12 Z
12 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

The **FN 18: SYSREAD** function can be used to read numeric system data and save the value in a Q, QL, or QR parameter (e.g., **FN 18: SYSREAD Q25 = ID210 NR4 IDX3**.)

> 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: "FN 18: SYSREAD – Reading system data", Page 243

The **SYSSTR** function can be used to read alphanumeric system data and save the value in a QS parameter (e.g., **QS25 = SYSSTR(ID 10950 NR1)**).

Further information: "Reading system data", Page 253

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
		8	1	Unit of measure of the directly calling NC program (may also be a cycle). Return codes: 0 = mm 1 = inch –1 = there is no corresponding program
			2	Unit of measure of the NC program visible in the block display from which the current cycle was called directly or indirectly. Return codes: 0 = mm 1 = inch –1 = there is no corresponding program
		9	-	Within an M function macro: Number of the M function. Otherwise –1
			-	Within an M function macro: Number of the M function. Otherwise –1
		10	-	Repeat counter: Indicates the number of times the current code has been executed since the current NC program call
		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 resolves 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	-	Number or name (string or QS) of the label to which the NC program will jump if **FN 14: ERROR** has been programmed with the NC CANCEL reaction, instead of aborting the NC program with an error message. The error number programmed in the **FN 14** command can be read under ID992 NR14. Value = 0: **FN 14** has a 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.
Indexed access to Q parameters				
	15	11	Q parameter number	Reads Q(IDX)
		12	QL parameter no.	Reads QL(IDX)
		13	QR parameter no.	Reads QR(IDX)
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		11	-	Index of active tool
		14	-	Number of active spindle
		20	-	Programmed cutting speed in turning operation
		21	-	Spindle mode in turning mode: 0 = constant speed 1 = constant cutting speed
		22	-	Coolant status M7: 0 = inactive, 1 = active
		23	-	Coolant status M8: 0 = inactive, 1 = active

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
		48	-	Tolerance
		49	-	HSC mode (Cycle 32 Tolerance)
		50	-	Tolerance for rotary axes (Cycle 32 Tolerance)
		52	Q parameter number	Type of transfer parameter for user cycles: –1: Cycle parameter not programmed in CYCL DEF 0: Cycle parameter numerically programmed in CYCL DEF (Q parameter) 1: Cycle parameter programmed as string in CYCL DEF (Q parameter)
		60	-	Clearance height (touch probe cycles 30 to 33)
		61	-	Inspection (touch probe cycles 30 to 33)
		62	-	Cutting edge measurement (touch probe cycles 30 to 33)
		63	-	Q parameter number for the result (touch probe cycles 30 to 33)
		64	-	Q parameter type for the result (touch probe cycles 30 to 33) 1 = Q, 2 = QL, 3 = QR
		70	-	Multiplier for feed rate (cycles 17 and 18)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modal status				
	35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
		2	-	Radius compensation: 0 = R0 1 = RR/RL 10 = Face milling 11 = Peripheral milling
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
		44	Tool no.	Exceeding the tool life
		45	Tool no.	Front-face width of indexable insert (RCUTS)
		46	Tool no.	Usable length of the milling cutter
		47	Tool no.	Neck radius of the milling cutter (RN)

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
File information				
	56	1	-	Number of lines of the tool table
		2	-	Number of lines of the active datum table
		4	-	Number of rows in a freely definable table that has been opened with **FN 26: TABOPEN**
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	-	Cutting speed [mm/min]
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	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 [kgm^2] (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 stage. 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 oversize	Active length

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			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 to 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 to 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.
		10	-	Type of definition of the active tilt: 0 = no tilt—is returned if, both in **Manual Operation** and in the automatic modes, no tilt is active. 1 = axial 2 = spatial angle
		11	-	Coordinate system for manual movements: 0 = Machine coordinate system **M-CS** 1 = Working plane coordinate system **WPL-CS** 2 = Tool coordinate system **T-CS** 4 = Workpiece coordinate system **W-CS**

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		12	Axis	Correction in working plane coordinate system **WPL-CS** (FUNCTION TURNDATA CORR WPL or FUNCTION CORRDATA WPL) Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)

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 to 3 (rotA, rotB, rotC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read OEM offset values.. Index: 1 to 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 to 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
Nominal positions of the physical axes in the REF system				
	245	1	Axis	Current nominal positions of the physical axes 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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read the current position in the active coordinate system, including offsets (handwheel, etc.)				
	271	1	Axis	Current nominal position in the input system
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)
		5	Secondary axis	Read whether the given secondary axis is used in the kinematics model. –1 = Axis not in the kinematics model 0 = Axis is not included in the kinematics calculation:
		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 to 9 (X, Y, Z, A, B, C, U, V, W)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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
		126	-	M126: –1 = on, 0 = off
Current system time				
	320	1	0	System time in seconds that have elapsed since 01.01.1970, 00:00:00 (real time).
			1	System time in seconds that have 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
		16	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

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: DD.MM.YYYY hh:mm
		20	0	The current calendar week number according to ISO 8601 (real time)
			1	The current calendar week number according to ISO 8601 (look-ahead calculation)
Global Program Settings (GPS): Global activation status				
	330	0	-	0 = No Global Program Settings active 1 = Any GPS settings active
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No Global Program Settings active 1 = Any GPS settings 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 to 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 to 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 to 10 (X, Y, Z, A, B, C, U, V, W, VT)
		9	Axis	GPS: Value for handwheel superimpositioning Index: 1 to 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 to 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 to 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
			3	TT: Designation of the active line in the touch-probe table
			4	TT: Touch probe input
		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
			-	TT: Stop probing movement upon stylus deflection
		100	-	Distance after which the probe is deflected during touch probe simulation

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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 3D 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
Preset from the touch probe cycle (probing results)				
	360	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
Preset from touch probe cycle (probing results)				
	360	10	-	Oriented spindle stop
		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
Settings for touch probe cycles				
	370	2	-	Rapid traverse for measurement
		3	-	Machine rapid traverse as rapid traverse for measurement
		5	-	Angle tracking on/off
		6	-	Automatic measuring cycles: interruption with info about on/off
Settings for touch-probe cycles				
	370	7	-	Reaction when the automatic 14xx measuring cycle does not reach the probing point: 0 = Cancellation 1 = Warning 2 = No message

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				In case of values 1 and 2, the measurement result must be evaluated, and a corresponding reaction is required.
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 to 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 to 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, then the function returns the value -1.
		2	-	Number of the active pallet preset. Same as NO1.
Values for the basic transformation of the pallet preset				
	547	Row number	Axis	Read the basic transformation values 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 the axis-offset values from the pallet preset table.. Index: 1 to 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset.. Index: 4 to 9 (A_OFFS, B_OFFS, C_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**)
		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**)
		31	Index of physical axis	Form (**MP_shape**) of the **axisCutterLoc** filter 0: Off 1: Average

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				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**)
		45	-	Shape of the smoothing filter (**CfgSmoothingFilter/shape**) 0 = Off 1 = Average 2 = Triangle
		46	-	Order of smoothing filter (only odd-numbered values) (**CfgSmoothingFilter/order**)
		47	-	Type of acceleration profile (**CfgLaPath/profileType**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		48	-	Type of acceleration profile for rapid traverse (**CfgLaPath/profileTypeHi**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal
		49	-	Filter reduction mode (**CfgPositionFilter/timeGainAtStop**) 0 = Off 1 = NoOvershoot 2 = FullReduction
		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**)
		53	Index of physical axis	Radial jerk, normal feed rate (**MP_maxTransJerk**)
		54	Index of physical axis	Radial jerk, high feed rate (**MP_maxTransJerkHi**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read or write look-ahead parameters of a single axis (at cycle level)				
	613	see ID610	see ID610	Same as ID610 but is only in effect at the cycle level. Overwrite values from the machine configuration and values at the machine level. **Further information:** "FN functions ID610, ID611, ID613", Page
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
		3	-	Read the SIK type (generation) 1 = SIK1 or no SIK 2 = SIK2
		4	Option number (4 digits)	Read the status of a software option (only available with SIK2) 0 = Not enabled 1 or higher = Number of enabled options
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC7, TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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
		45	-	Front-face width of indexable insert (RCUTS)
		46	-	Usable length of the milling cutter
		47	-	Neck radius of the milling cutter (RN)
		48	-	Radius at the tool tip (R_TIP)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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 NO1
Touch probe cycles and coordinate transformations				
	990	1	-	Approach behavior: 0 = Standard behavior 1 = Approach probing position without compensation. Effective radius, set-up clearance is zero
		2	16	Automatic / Manual machine operating modes
		4	-	0 = Stylus not deflected 1 = Stylus deflected
		6	-	TT tool touch probe active? 1 = Yes 0 = No
		8	-	Momentary spindle angle in [°]
		10	QS parameter no.	Determine the tool number from the tool name. The return value depends on the rules configured for the search of the replacement tool. If there are multiple tools with the same name, the first tool from the tool table will be selected. If the tool selected by these rules is locked, a replacement tool will be returned. –1: No tool with the specified name found in the tool table or all qualifying tools are locked.
		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 (CfgMa-

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				chineSimul/simMode parameter = FullOperation, can be programmed for testing purposes)
		28	-	Read inclination angle of the current tool spindle

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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 **FN 14** error
		16	-	Real execution active? 1 = execution, 0 = simulation
		17	-	2D graphics during programming active? 1 = Yes 0 = No
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		32	Cycle number	Single cycle enabled: 0 = No 1 = Yes
		33	-	Write-access enabled for DNC (Python scripts) for executed entries in the pallet table: 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 the **Spindle is not rotating** error message (CfgGeoCycle/**displaySpindleErr**) 0 = No, 1 = Yes
		2	-	Display the **Check the depth sign** error message (CfgGeoCycle/**displayDepthErr**) 0 = No, 1 = Yes
Data transfer between HEIDENHAIN cycles and OEM macros				
	1031	1	0	Component monitoring: counter of the measurement. Cycle 238 Measure machine data automatically increments this counter.
			1	Component monitoring: Type of measurement –1 = No measurement 0 = Circular interpolation test 1 = Waterfall chart test 2 = Frequency response 3 = Envelope curve spectrum 4 = Advanced frequency response
			2	Component monitoring: Index of the axis from CfgAxes**axisList**
			3 – 9	Component monitoring: further arguments depend on the measurement
		2	3 – 9	Component monitoring: further arguments depend on the measurement
		3	0	KinematicsOpt: Read the current cycle number (450-453)
		100	-	Component monitoring: optional names of the monitoring tasks, as specified in **System \Monitoring\CfgMonComponent**. After completion of the measurement, the monitoring tasks stated here are executed consecutively. When assigning the input parameters, remember to separate the listed monitoring tasks by commas.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
User settings for the user interface				
	1070	1	-	Feed rate limit of soft key FMAX; 0 = FMAX is inactive
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 large 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 "..."**.
Indexed access to QS parameters				
	10015	20	QS parameter no.	Reads QS(IDX)
		30	QS parameter no.	Returns the string that you obtain if you replace anything except for letters and digits in QS(IDX) by '_'.
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
		13	-	Symbolic name of the grinding tool table
		14	-	Symbolic name of the dressing tool table
		21	-	Symbolic name of the compensation table in the T-CS tool coordinate system
		22	-	Symbolic name of the compensation table in the WPL-CS working plane coordinate system

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	0 - 16, 20	-	1: DD.MM.YYYY hh:mm:ss 2: D.MM.YYYY h:mm 3: DD.MM.YY hh:mm 4: YYYY-MM-DD hh:mm:ss 5: YYYY-MM-DD hh:mm 6: YYYY-MM-DD h:mm 7: YY-MM-DD h:mm 8: DD.MM.YYYY 9: D.MM.YYYY 10: D.MM.YY 11: YYYY-MM-DD 12: YY-MM-DD 13: hh:mm:ss 14: h:mm:ss 15: h:mm 16: DD.MM.YYYY hh:mm 20: Calender week as per ISO 8601 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	-	Type of TS probe from TYPE column of the touch probe table (**tchprobe.tp**)
		51	-	Shape of stylus from column STYLUS in 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**.
		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)				

Group name	Group number ID...	System data number NO....	Index IDX...	Description
	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
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
Read current tool data (system string)				
	10950	6	-	Entry from the TSHAPE column - file name of the 3D tool shape (*.stl)
Read information from OEM macros and HEIDENHAIN cycles (system string)				
	11031	10	-	Returns the selection of the FUNCTION MODE SET <OEM mode> macro as a string.
		100	-	Cycle 238: list of key names for component monitoring
		101	-	Cycle 238: file names for log file

Comparison: FN 18 functions

The following table lists the FN 18 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)]

No.	IDX	Contents	Replacement function
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)
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 no.	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 FN 26: 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	

No.	IDX	Contents	Replacement function
8	-	Tolerance programmed with Cycle 32	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
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			

No.	IDX	Contents	Replacement function
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	
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 and FN 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 and FN 28 or SQL to read out the table cell

18.2 Technical Information

Specifications

Explanation of symbols

■ Standard function
□ Axis option
1 Advanced Function Set 1
x Software option, except Advanced Function Set 1 and Advanced Function Set 2

Specifications	
Components	■ Main computer ■ Operating panel ■ Screen with soft keys
Program memory	■ 2 GB
Input resolution and display step	■ As fine as 0.1 µm for linear axes ■ As fine as 0.000 1° 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/4096 ■ Position controller cycle time: 200 µs (100 µs with option 49) ■ Speed controller cycle time: 200 µs (100 µs with option 49) ■ Current controller cycle time: minimum 100 µs (minimum 50 µs with option 49)
Spindle speed	■ max. 100 000 rpm (with 2 pole pairs)
Error compensation	■ Linear and nonlinear axis errors, backlash, thermal expansion ■ Static friction, sliding friction

Specifications		
Data interfaces	■	One each: RS-232-C/V.24 max. 115 kbit/s
	■	Extended data interface with LSV-2 protocol for remote operation of the control through the data interface with the TNCremo or TNCremoPlus software
	■	2 x Gigabit Ethernet interface 1000BASE-T
	■	3 USB ports: 1 front USB 2.0 port and 2 rear USB 3.0 ports)
	x	HEIDENHAIN-DNC for communication between a Windows application and TNC (DCOM interface)
	x	OPC UA NC Server Stable and reliable interface for the connection of leading-edge industrial applications
Ambient temperature	■	Operation: +5 °C to +45 °C
	■	Storage: -20 °C to +60 °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 in **TOOL CALL** block. 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	–99.9999 to +99.9999 (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 9999 (4, 0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	–999 999 999.999999 to +999 999 999.999999 (9, 6)
Labels (LBL) for program jumps	0 to 65535 (5, 0)
Labels (LBL) for program jumps	Any text string in quotes (“”)
Number of program-section repeats REP	1 to 65 534 (5, 0)
Error number for Q parameter function FN 14	0 to 1199 (4, 0)

User functions

User functions	Standard	Option	Meaning
Short description	✓		Basic version: 3 axes plus closed-loop spindle
		0	1st additional axis for 4 axes plus closed-loop spindle
		1	2nd additional axis for 5 axes plus closed-loop spindle
Program entry			In HEIDENHAIN Klartext 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
Cutting data	✓		Automatic calculation of spindle speed, cutting speed, feed per tooth, and feed per revolution
Program jumps	✓		Subprograms
	✓		Program section repeats
	✓		External NC programs
Machining cycles	✓		Cycles for drilling, and conventional and rigid tapping
		19	Cycles for pecking, reaming, boring, and counterboring
	✓		Roughing and finishing rectangular pockets
	✓		Roughing and finishing rectangular studs
	✓		Cycles for clearing level surfaces
	✓		Face milling
	✓		Cartesian and polar point patterns
	✓		OEM cycles (special cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	✓		Datum shift, mirroring
	✓		Scaling factor (axis-specific)
Q parameters Programming with variables	✓		Mathematical functions =, +, –, *, /, roots
	✓		Logical operations (=, ≠, <, >)
	✓		Calculating with parentheses
	✓		sin α, cos α, tan α , arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point
	✓		Functions for calculation of circles
	✓		String parameters

User functions	Standard	Option	Meaning
Programming aids	✓		Calculator
	✓		Color highlighting of syntax elements
	✓		Complete list of all current error messages
	✓		Context-sensitive help function
	✓		Graphic support for the programming of cycles
	✓		Comment and structure blocks in the NC program
Teach-In	✓		Actual positions can be transferred directly to the NC program
Test graphics	✓		Graphic simulation before a program run, even while another NC program is being run
Display modes	✓		Plan view / projection in 3 planes / 3D 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 (2D 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 / 3D 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 datums
Returning to the contour	✓		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
	✓		Manual presetting
	✓		Tools can be measured automatically

For a detailed overview of the user functions, see the brochure for the TNC 128. You can find the brochures related to the product range of CNC controls in the download area of the HEIDENHAIN website.

Software options

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

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

Accessories

Accessories	
Electronic handwheels	■ HR 510: Portable handwheel ■ HR 550FS: Portable radio handwheel with display ■ HR 520: 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: workpiece touch trigger probe with cable connection ■ TS 260: workpiece touch trigger probe with cable connection ■ TT 160: tool touch trigger probe ■ KT 130: Simple touch trigger probe with cable connection

Fixed cycles

Cycle number	Cycle name	DEF active	CALL active
7	DATUM SHIFT	■	
8	MIRRORING	■	
9	DWELL TIME	■	
11	SCALING FACTOR	■	
12	PGM CALL		■
13	ORIENTATION	■	
26	AXIS-SPECIFIC SCALING	■	
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		■	169
M1	Optional program STOP/Spindle STOP/Coolant OFF		■	169
M2	Program STOP/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 0		■	169
M3 M4 M5	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	■ ■ 	 ■	169
M8 M9	Coolant ON Coolant OFF	■ 	 ■	169
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON	■ ■		169
M30	Same function as M2		■	169
M89	Cycle call, modally effective	■ 	 ■	351
M91	Within the positioning block: Coordinates are referenced to machine datum	■		170
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position	■		170
M94	Reduce the rotary axis display to a value below 360°	■		172
M99	Blockwise cycle call		■	351
M103	Feed rate factor for plunging movements	■		173
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136	■		174
M140	Retraction from the contour in the tool-axis direction	■		174

Index

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.

For more details on touch probes and vision systems:
www.heidenhain.com/products/touch-probes-and-vision-systems

Original document
819494-26 · Ver06 · SW18 · 10/2023 · H · Printed in Germany