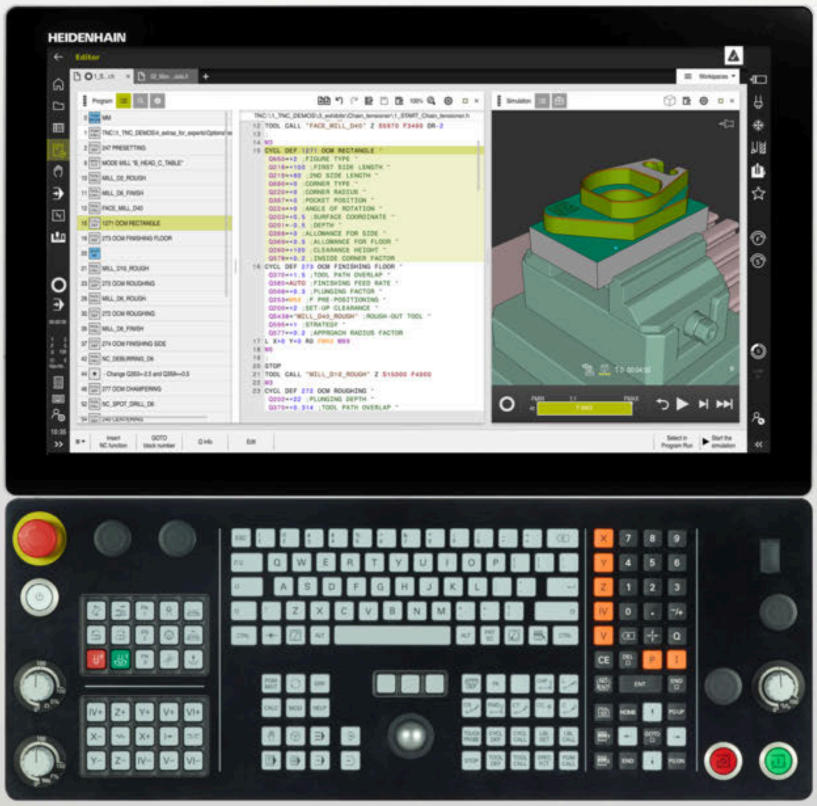
HEIDENHAIN



TNC7

User's Manual
Complete Edition

NC Software
81762x-17

English (en)
10/2022

Contents

1

New and Modified Functions

New functions 81762x-17

- You can run and edit ISO programs.

  Further information: "ISO", Page 1477
- In the text editor mode, the control provides an auto-complete function when programming. The control suggests syntax elements matching your entries which you can apply to the NC program.

  Further information: "Inserting NC functions", Page 228
- If an NC block contains a syntax error, the control displays a symbol in front of the block number. When you select the symbol, the control displays the corresponding error description.

  Further information: "Editing NC functions", Page 230
- In the **Klartext** area of the **Program settings** window, you select whether the control skips the offered optional syntax elements of an NC block during input.

  If the toggle switches in the **Klartext** area are active, the control skips the syntax elements Comment, Tool index and Linear superimposition.

  Further information: "Settings in the Program workspace", Page 220
- If the control does not process or simulate the miscellaneous function **M1** or NC blocks hidden with **/**, it then grays out the miscellaneous function or NC blocks.

  Further information: "Appearance of the NC program", Page 219
- When programming circular paths with **C**, **CR** and **CT**, the **LIN_** syntax element is now available in order to superimpose a linear motion over the circular motion of an axis. This allows you to program a helix in a simple way.

  In ISO programs, you can define a third axis in conjunction with the **G02**, **G03**, and **G05** functions.

  Further information: "Linear superimpositioning of a circular path", Page 341
- You can save up to 200 successive NC blocks as NC sequences and insert them during programming using the **Insert NC function** window. In contrast to the called NC programs, you can adapt the NC sequences after insertion without changing the actual sequence.

  Further information: "NC sequences for reuse", Page 392
- The **FN 18: SYSREAD** (ISO: **D18**) functions have been enhanced:
  - **FN 18: SYSREAD (D18) ID610 NR49**: Mode of filter reduction of one axis (**IDX**) for **M120**
  - **FN 18: SYSREAD (D18) ID780**: Information on the current grinding tool
    - **NR60**: Active compensation method in **COR_TYPE** column
    - **NR61**: Inclination angle of dressing tool
  - **FN 18: SYSREAD (D18) ID950 NR48**: Value in column **R_TIP** in the tool table for the current tool
  - **FN 18: SYSREAD (D18) ID11031 NR101**: File name of the log file of Cycle **238 MEASURE MACHINE STATUS**

  Further information: "System data", Page 2271

- In the **Visualization options** column of the **Simulation** workspace, you can show the worktable and, if necessary, the fixtures, in **Workpiece** mode and with the **Clamping situation** toggle switch.

  Further information: "Visualization options column", Page 1537
- In the context menu of the **Editor** operating mode and the **MDI** application, the control offers the **Insert last NC block** function. With this function you can insert the last deleted or edited NC block in any NC program.

  Further information: "Context menu in the Program workspace", Page 1525

- You can perform file functions in the **Save as** window using the context menu.
  Further information: "Context menu", Page 1522
- When you add a favorite or lock a file in the file management, the control displays an icon next to the file or folder.
  Further information: "Basic information", Page 1144
- The **Document** workspace has been added. In the **Document** workspace, you can open files in order to view them, such as a technical drawing.
  Further information: "Document workspace", Page 1154
- Software option 159 (Model Aided Setup) has been added.
  This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions.
  The control supports you additionally by showing the clamping situation and possible touch points in the **Simulation** workspace by means of a 3D model.
  Further information: "Setting up the workpiece with graphical support (option 159)", Page 1582
- If you execute an NC program or a pallet table or if you test it in the opened **Simulation** workspace, the control displays a navigation path in the file information bar of the **Program** workspace. The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when the program run is interrupted.
  Further information: "Navigation path in the Program workspace", Page 1962
- The **TRANS** tab of the **Status** workspace indicates the active shift in the working plane coordinate system **WPL-CS**. If the shift comes from a compensation table (***.WCO**), the control shows the path to the compensation table as well as the number and, if applicable, the comment of the active row.
  Further information: "TRANS tab", Page 181
- You can transfer tables from earlier control models to the TNC7. If columns are missing in the table, the control opens the **Incomplete table layout** window.
  Further information: "Tables operating mode", Page 1980
- The **Form** workspace in the **Tables** operating mode has been expanded:
  - The control displays an icon of the selected tool type in the **Tool Icon** area. For the turning tools the icons also take into account the tool orientation and show where the relevant tool data will apply.
  - Use the up and down arrows in the title bar to select the previous or next table row.

  Further information: "Form workspace for tables", Page 1989
- You can create user-defined filters for the tool tables and pocket table. To do this, define a search condition in the **Search** column which you save as a filter.
  Further information: "Search column in the Table workspace", Page 1986

- The following tool types have been added:
  - **Face mill** (**MILL_FACE**)
  - **Chamfer cutter** (**MILL_CHAMFER**)

  Further information: "Tool types", Page 281
- 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.

  Further information: "Database ID", Page 276
- You define a radius at the tip of the tool in the **R_TIP** column of the tool table.

  Further information: "Tool table tool.t", Page 1995
- 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.

  Further information: "Touch probe table tchprobe.tp", Page 2022
- You define the compensation method for dressing in the **COR_TYPE** input parameter for grinding tools (option 156):
  - **Grinding wheel with compensation, COR_TYPE_GRINDTOOL**

    Stock removal on the grinding tool
  - **Dressing tool with wear, COR_TYPE_DRESSTOOL**

    Stock removal on dressing tool

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010
- Each user can create and activate configurations in which the control's user interface is individually adapted.

  You can save and activate individual modifications to the control's user interface as a configuration, e.g. for each operator. The configuration contains, for example, favorites and the arrangement of the workspaces.

  Further information: "Configuring the control's user interface", Page 2157
- The **OPC UA NC Server** enables client applications to access the tool data of the control. You can read and write tool data.

  The **OPC UA NC Server** does not provide access to the grinding and dressing tool tables (option 156).

  Further information: "OPC UA NC Server (options 56 to 61)", Page 2123
- Use the machine parameter **stdTNChelp** (no. 105405) to define whether the control displays help graphics as pop-up windows in the **Program** workspace.
- The optional machine parameter **CfgGlobalSettings** (no. 128700) allows you to define whether the control offers the parallel axes for **Handwheel superimp.**.

  Further information: "Function Handwheel superimp.", Page 1225

New cycle functions with 81762x-17

- Cycle **1416 INTERSECTION PROBING** (ISO: **G1416**)

  This cycle allows you to determine the intersection of two edges. The cycle requires a total of four touch points and two positions per edge. You can use the cycle in the three object planes **XY**, **XZ** and **YZ**.

  Further information: "Cycle 1416 INTERSECTION PROBING", Page 1641
- Cycle **1404 PROBE SLOT/RIDGE** (ISO: **G1404**)

  This cycle determines the center and the width of a slot or ridge. The control probes two opposing touch points. You can also define a rotation for the slot or the ridge.

  Further information: "Cycle 1404 PROBE SLOT/RIDGE ", Page 1690
- Cycle **1430 PROBE POSITION OF UNDERCUT** (ISO: **G1430**)

  This cycle determines a single position with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus.

  Further information: "Cycle 1430 PROBE POSITION OF UNDERCUT ", Page 1695
- Cycle **1434 PROBE SLOT/RIDGE UNDERCUT** (ISO: **G1434**)

  This cycle determines the center and the width of a slot or ridge with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes two opposing touch points.

  Further information: "Cycle 1434 PROBE SLOT/RIDGE UNDERCUT ", Page 1700

Changed functions 81762x-17

- If you press the **actual position capture** key in the **Editor** operating mode or the **MDI** application, the control creates a straight line **L** with the current position of all axes.
- If you select the tool with the selection window when calling the tool with **TOOL CALL**, you can switch via an icon to the **Tables** operating mode. In this case, the control displays the selected tool in the **Tool management** application.

  Further information: "Tool call by TOOL CALL", Page 304
- You can use the **TABDATA** functions for read- and write-access to the preset table.

  Further information: "Accessing table values ", Page 1991
- If you define a grinding tool (option 156) with orientation **9** or **10**, the control supports circumferential milling in conjunction with **FUNCTION PROG PATH IS CONTOUR** (option 9).

  Further information: "3D tool compensation with the entire tool radius with FUNCTION PROG PATH (option 9)", Page 1139
- When you save an input value, the control removes superfluous zeros at the beginning of the input and at the end of the decimal places. The input range must not be exceeded for this.
- The control no longer interprets tab characters as syntax errors. In comments and structure items, the control displays a tab character as a space. In syntax elements, the control removes a tab character.
- If you edit a value and press the backspace key, the control deletes only the last character and not the complete input.
- You can delete an empty line with the backspace key in text editor mode.
- The **Insert NC function** window has been expanded:
  - In the areas **Search result**, **Favorites** and **Last functions**, the control shows the path of the NC functions.
  - If you select an NC function and swipe to the right, the control displays the following file functions:
    - Add to or remove from favorites
    - Open containing folder

      Only when you search for an NC function
  - If software options are not enabled, the control shows unavailable contents in the **Insert NC function** window grayed out.

  Further information: "Inserting NC functions", Page 228
- Graphical programming has been enhanced:
  - If you select the face of a closed contour, you can insert a radius or chamfer at each corner of the contour.
  - In the Element Information area, the control shows a rounding arc as **RND** contour element and a chamfer as **CHF** contour element.

  Further information: "Controls and gestures in graphical programming", Page 1439

- For a screen output with **FN 16: F-PRINT** (ISO: **D16**), the control displays a pop-up window.

  Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1382
- The window **Q parameter list** contains an input field that allows you to navigate to a unique variable number. If you press the **GOTO** key, the control selects the input field.

  Further information: "Q parameter list window", Page 1366
- The structure of the **Program** workspace has been enhanced:
  - The structure contains the NC functions **APPR** and **DEP** as structure elements.
  - The control shows comments in the structure inserted within structure elements.
  - If you mark structuring items in the **Structure** column, the control propagates the marking to the corresponding NC blocks in the NC program. Use the **CTRL+SPACE** key shortcut to stop marking. If you press **CTRL+SPACE** again, the control restores the marked selection.

  Further information: "Structure column in the Program workspace", Page 1514
- The **Search** column in the **Program** workspace has been enhanced:
  - The **Match whole words only** checkbox determines that the control shows only exact matches. If, for example, you search for **Z+10**, the control ignores **Z+100**.
  - If in the **Search and replace** function you use **Find next**, the control highlights the first result in purple.
  - If you do not enter a value for **Replace with:**, the control deletes the value searched for and to be replaced.

  Further information: "Search column in the Program workspace", Page 1517
- If you select several NC blocks during the program comparison, you can load all NC blocks simultaneously.

  Further information: "Program comparison", Page 1520
- The control provides additional keyboard shortcuts to mark NC blocks and files.
- When you open or save a file in a selection window, the control displays the context menu.

  Further information: "Context menu", Page 1522
- The cutting data calculator has been enhanced:
  - You can load the tool name from the cutting data calculator.
  - If you press the enter key in the cutting data calculator, the control selects the next element.

  Further information: "Cutting data calculator", Page 1529

- The **Workpiece position** window of the **Simulation** workspace has been enhanced:
  - You can use a button to select a workpiece preset from the preset table.
  - The control displays the input fields below each other instead of next to each other.

  Further information: "Visualization options column", Page 1537
- The control can display a finished part in the **Machine** mode of the **Simulation** workspace.

  Further information: "Workpiece options column", Page 1539
- The control takes into account the following columns of the tool table for the simulation:
  - **R_TIP**
  - **LU**
  - **RN**

  Further information: "Simulation of tools", Page 1544
- In the Simulation function of the **Editor** operating mode, the control takes dwell times into account. The control does not dwell during the program test, but adds the dwell times to the program run time.
- The NC functions **FUNCTION FILE** and **FN 27: TABWRITE** (ISO: **D27**) are active in the **Simulation** workspace.

  Further information: "Simulation Workspace", Page 1535
- File management has been enhanced:
  - The control shows the occupied memory and total memory of the drives in the navigation bar of the file management.
  - The control shows STEP files in the preview area.

    Further information: "Screen elements of the file management", Page 1146
  - When you cut a file or folder in the file management, the control grays out the icon of the file or folder.

    Further information: "Icons and buttons", Page 1144
- The **Quick selection** workspace has been enhanced:
  - Tables for execution and simulation can be opened in the **Quick selection** workspace in the **Tables** operating mode.
  - In the **Quick selection** workspace in the **Editor** operating mode, you can create NC programs with mm or inch units of measurement as well as ISO programs.

  Further information: "Quick selection workspace", Page 1154
- If you check the pallet table in Batch Process Manager (option 154) with Dynamic Collision Monitoring (DCM, option 40), the control takes the software limit switches into account.

  Further information: "Batch Process Manager (option 154)", Page 1943

- If you shut down the control with still unsaved changes in NC programs and contours, the control displays the **Close the program** window. You can save the changes, discard them or cancel the shutdown.

  Further information: "Powering off", Page 198
- You can change the size of windows. The control remembers the size until it is shut down.

  Further information: "Icons on the control's user interface", Page 123

- In the **Files**, **Tables** and **Editor** operating modes, a maximum of ten tabs can be open at the same time. If you try to open additional tabs, the control shows a message.

  Further information: "Areas of the control's user interface", Page 109
- **CAD-Viewer** has been enhanced:
  - Internally, **CAD-Viewer** always uses mm for its calculations. If you select inches as the unit of measure, **CAD-Viewer** will convert 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. In 2D mode, the control grays out the Z coordinate.
  - **CAD-Viewer** also recognizes circles that consist of two semi-circles as machining positions.
  - You can save the information on the workpiece preset and workpiece datum to a file or to the clipboard without having to resort to CAD Import (software option 42).

  Further information: "Opening CAD Files with the CAD-Viewer", Page 1455
- The **Open in the editor** button in the **Program Run** operating mode opens the currently displayed NC program, including called NC programs.

  Further information: "Program Run operating mode", Page 1954
- In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

  Further information: "Manual traverse during an interruption", Page 1964
- Process monitoring (option 168) has been enhanced:
  - The **Process Monitoring** workspace contains a setup mode. When the mode is inactive, the control hides all functions for setting up process monitoring.

    Further information: "Icons", Page 1239
  - When the settings of a monitoring task are selected, the control shows two areas with the original settings and the current settings of the monitoring task.

    Further information: "Monitoring tasks", Page 1245
  - The control shows the coverage, i.e. the matching of the current graph with the graph of the reference machining, as circular charts.

    The control shows reactions of the notification menu in the graph and in the table of recordings.

    Further information: "Records of monitoring sections", Page 1257

- The status overview on the TNC bar has been enhanced:
  - In the status overview, the control displays the run time of the NC program in mm:ss format. As soon as the run time of the NC program exceeds 59:59, the control shows the run time in hh:mm format.
  - If a tool usage file is available, the control calculates for the **Program Run** operating mode how long the execution of the active NC program will take. During program run the control updates the remaining run time. The control shows the remaining run time in the status overview on the TNC bar.
  - If more than eight axes are defined, the control shows the axes in two columns in the position display of the status overview. With more than 16 axes, the control shows the axes in three columns.

  Further information: "Status overview on the TNC bar", Page 169
- The control shows a feed rate limit in the status display:
  - If a feed rate limit is active, the control highlights the **FMAX** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the control shows the feed rate in orange.
  - If the feed rate is limited using the **FMAX** button, the control displays **MAX** in square brackets.

    Further information: "Feed rate limit FMAX", Page 1958
  - If the feed rate is limited using the **F limited** button, the control displays the active safety function in square brackets.

    Further information: "Safety functions", Page 2092
- In the **Tool** tab of the **Status** workspace, the control displays the values of the **Tool geometry** and **Tool allowances** areas with four instead of three decimal places.

  Further information: "Tool tab", Page 184
- 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.

  Further information: "Contents of an electronic handwheel display", Page 2072

- If you align the rotary table after a manual touch probe function, the control remembers the selected type of rotary axis positioning and the feed rate.

  Further information: "Buttons", Page 1562
- If you correct the preset or datum after a manual touch probe function, the control shows a symbol behind the adopted value.

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557
- In the **3-D rotation** window (option 8), if you enable a function in the **Manual Operation** or **Program run** areas, the control highlights the area in green.

  Further information: "3-D rotation window (option 8)", Page 1098
- The **Tables** operating mode has been enhanced:
  - The **M** and **S** statuses are highlighted in color only for the active application, and gray for the other applications.
  - You can close all applications except for **Tool management**.
  - The **Mark row** button has been added.
  - In the **Presets** application, the **Lock record** toggle switch has been added.

  Further information: "Tables operating mode", Page 1980
- The **Table** workspace has been enhanced:
  - You can change the column width using an icon.
  - In the settings of the **Table** workspace you can enable or disable all table columns and restore the default format.

  Further information: "Table workspace", Page 1982
- If a table column offers two input options, the control shows the options in the **Form** workspace as toggle switches.
- The minimum input value of the **FMAX** column in the touch probe table has been changed from –9999 to +10.

  Further information: "Touch probe table tchprobe.tp", Page 2022
- You can import tool tables of the TNC 640 as CSV files.

  Further information: "Importing tool data", Page 299

- The maximum input range of the **LTOL** and **RTOL** columns of the tool table has been increased. It was from 0 to 0.9999 mm, and is now from 0.0000 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 to 3.2767 mm, and is now from 0.0000 to 9.0000 mm.

  Further information: "Tool table tool.t", Page 1995
- If you double tap or click a tool in the **Tool check** column of the **Program** workspace, the control switches to the **Tables** operating mode. In this case, the control displays the selected tool in the **Tool management** application.

  Further information: "Tool check column in the Program workspace", Page 313
- In the expanded notification menu, the control displays information about the NC program in a separate area outside of the **Details**.

  Further information: "Message menu on the information bar", Page 1532
- You can use the **Update documentation** function to install or update, for example, the **TNCguide** integrated product aid.

  Further information: "Update documentation", Page 2150
- The control no longer supports the ITC 750 additional operating station.
- When you enter a code number in the **Settings** application, the control displays a load icon.

  Further information: "Code numbers", Page 2103
- In the **DNC** menu item of the **Settings** application, the **Secure connections for users** area has been added. These functions can be used to define settings for secure connections via SSH.

  Further information: "Secure connections for user", Page 2129
- 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 needing to transmit them to the control.

  Further information: "SSH-secured DNC connection", Page 2178
- You can export and import existing network configurations in the **Network settings** window.

  Further information: "Exporting and importing a network profile", Page 2122
- The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).

  When the control detects a non-secure connection, it displays an informational notice.
- The optional machine parameter **warningAtDEL** (no. 105407) is used to define whether the control shows a confirmation request in a pop-up window when deleting an NC block.

Modified cycle functions with 81762x-17

- You can edit and execute Cycle **19 WORKING PLANE** (ISO: **G80**, option 8), but you cannot insert it into an NC program as a new element.
- Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167) monitors contour damage on the floor caused by the tool tip. This tool tip results from the radius **R**, the radius at the tool tip **R_TIP**, and the point angle **T-ANGLE**.

  Further information: "Cycle 277 OCM CHAMFERING (option 167) ", Page 683
- The parameter **Q592 TYPE OF DIMENSION** has been added to Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96). This parameter is used to define whether the contour is programmed with radius dimensions or diameter dimensions.

  Further information: "Cycle 292 CONTOUR.TURNG.INTRP. (option 96)", Page 694
- The following cycles consider the miscellaneous functions **M109** and **M110**:
  - Cycle **22 ROUGH-OUT** (ISO: G122)
  - Cycle **23 FLOOR FINISHING** (ISO: G123)
  - Cycle **24 SIDE FINISHING** (ISO: G124)
  - Cycle **25 CONTOUR TRAIN** (ISO: G125)
  - Cycle **275 TROCHOIDAL SLOT** (ISO: G275)
  - Cycle **276 THREE-D CONT. TRAIN** (ISO: G276)
  - Cycle **274 OCM FINISHING SIDE** (ISO: G274, option 167)
  - Cycle **277 OCM CHAMFERING** (ISO: G277, option 167)
  - Cycle **1025 GRINDING CONTOUR** (ISO: G1025, option 156)

  Further information: "SL cycles", Page 621

  Further information: "OCM cycles", Page 656

  Further information: "Cycle 1025 GRINDING CONTOUR (option 156)", Page 957
- If KinematicsComp (software option 52) is active, the log of Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48) shows the active compensations of the angular position errors (**locErrA**/**locErrB**/**locErrC**).

  Further information: "Cycle 451 MEASURE KINEMATICS (option 48)", Page 1873
- The log of Cycles **451 MEASURE KINEMATICS** (ISO: **G451**) and **452 PRESET COMPENSATION** (ISO: **G452**, option 48) contains diagrams with the measured and optimized errors of the individual measuring positions.

  Further information: "Cycle 451 MEASURE KINEMATICS (option 48)", Page 1873

  Further information: "Cycle 452 PRESET COMPENSATION (option 48)", Page 1888
- Cycle **453 KINEMATICS GRID** (ISO: **G453**, option 48) allows you to use the mode **Q406=0** even without KinematicsComp (software option 52).

  Further information: "Cycle 453 KINEMATICS GRID ", Page 1899
- Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**) determines the radius and, if required, the length, the center offset and the spindle angle of an L-shaped stylus.

  Further information: "Cycle 460 CALIBRATION OF TS ON A SPHERE (option 17)", Page 1858
- Cycles **444 PROBING IN 3-D** (ISO: **G444**) and **14xx** support probing with an L-shaped stylus.

  Further information: "Working with an L-shaped stylus", Page 1591

2

About the User's Manual

2.1 Target group: Users

A user is anyone who uses the control to perform at least one of the following tasks:

- Operating the machine
  - Setting up tools
  - Setting up workpieces
  - Machining workpieces
  - Eliminating possible errors during program run
- Creating and testing NC programs
  - Creating NC programs at the control or externally using a CAM system
  - Using the Simulation mode to test the NC programs
  - Eliminating possible errors during program test

The depth of information in the User's Manual results in the following qualification requirements on the user:

- Basic technical understanding, e.g. ability to read technical drawings and spatial imagination
- Basic knowledge in the field of metal cutting, e.g. meaning of material-specific parameters
- Safety instructions, e.g. possible dangers and their avoidance
- Training on the machine, e.g. axis directions and machine configuration

> HEIDENHAIN offers separate information products for other target groups:
>
> - Leaflets and overview of the product portfolio for potential buyers
> - Service Manual for service technicians
> - Technical Manual for machine manufacturers
>
> Additionally, HEIDENHAIN provides users and lateral entrants with a wide range of training opportunities in the field of NC programming.
> **HEIDENHAIN training portal**

In line with the target group, this User's Manual only contains information on the operation and use of the control. The information products for other target groups contain information on further product life phases.

2.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as User's Manual, regardless of the output or transport medium. Well-known designations with the same meaning include operator’s manual and operating instructions.

The User's Manual for the control is available in the variants below:

- As a printed version, sub-divided into the modules below:
  - The **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for running NC programs.
    ID: 1358774-xx
  - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe and machining cycles are not included.
    ID for Klartext programming: 1358773-xx
  - The **Machining Cycles** User's Manual contains all functions of the machining cycles.
    ID: 1358775-xx
  - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions of the touch probe cycles.
    ID: 1358777-xx
- As PDF files, sub-divided according to the printed versions or as a **Complete edition** User's Manual, containing all modules
  ID: 1369999-xx
  TNCguide
- As an HTML file used as the **TNCguide** product aid integrated directly into the control.
  TNCguide

The User's Manual supports you in the safe handling of the control according to its intended use.
Further information: "Proper and intended use", Page 89

Further information products for users

The following information products are available to you:

- **Overview of new and modified software functions** informs you about the innovations of specific software versions.
  TNCguide
- **HEIDENHAIN brochures** inform you about products and services by HEIDENHAIN, e.g. software options of the control.
  HEIDENHAIN brochures
- The **NC solutions** database offers solutions for frequently occurring tasks.
  HEIDENHAIN NC solutions

2.3 Types of notes used

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 contain the following four sections:

- Signal word indicating the hazard severity
- Type and source of hazard
- Consequences of ignoring the hazard for example: "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.

2.4 Notes on using NC programs

NC programs contained in this User's Manual are suggestions for solutions. The NC programs or individual NC blocks must be adapted before being used on a machine.

Change the following contents as needed:

- Tools
- Cutting parameters
- Feed rates
- Clearance height or safe position
- Machine-specific positions, e.g. with **M91**
- Paths of program calls

Some NC programs depend on the machine kinematics. Adapt these NC programs to your machine kinematics before the first test run.

In addition, test the NC programs using the simulation before the actual program run.

	With a program test you determine whether the NC program can be used with the available software options, the active machine kinematics and the current machine configuration.

2.5 User's Manual as integrated product aid: TNCguide

Application

The integrated product aid **TNCguide** offers the full content of all User's Manuals.

Further information: "Available user documentation", Page 79

The User's Manual supports you in the safe handling of the control according to its intended use.
Further information: "Proper and intended use", Page 89

Requirement

In the factory default setting, the control offers the integrated product aid **TNCguide** in German and English language versions.

If the control cannot find a **TNCguide** language version matching the selected dialog language, it opens **TNCguide** in English.

If the control cannot find a **TNCguide** language version, it opens an information page with instructions. With the link available there and the steps provided, you can supplement the files missing in the control.

> You can also open the information page manually by selecting the **index.html** file (for example, at **TNC:\tncguide\en\readme**). The path depends on the desired languageversion (e.g., **en** for English).
>
> With the steps provided you can also update the **TNCguide** version. Updating may be required (e.g., after a software update).

Description of function

The integrated product aid **TNCguide** can be selected within the **Help** application or in the **Help** workspace.

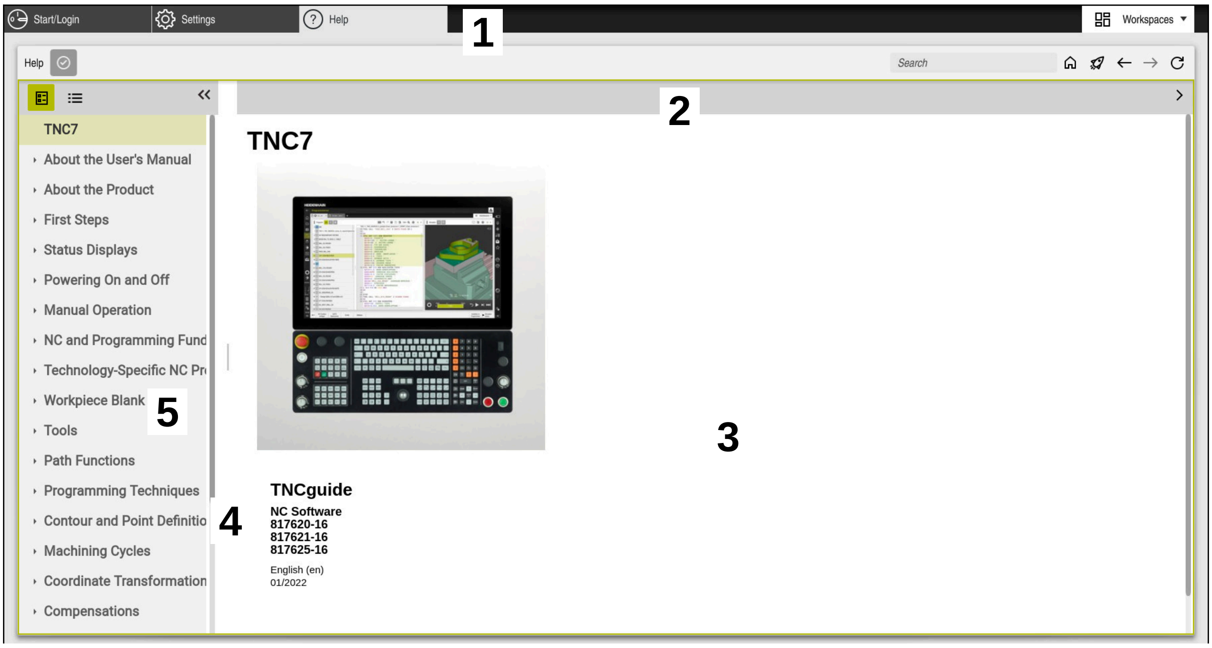
Further information: "Help application", Page 83

Further information: "Help workspace", Page 1506

Operation of **TNCguide** is identical in both cases.

Further information: "Symbols", Page 84

Help application



Help application with open **TNCguide**

The **Help** application includes the following areas:

1 Title bar in the **Help** application
Further information: "Symbols in the Help application", Page 84
2 Title bar of the integrated product aid **TNCguide**
Further information: "Symbols in the integrated product aid TNCguide", Page 84
3 Content column of **TNCguide**
4 Separator between the columns of **TNCguide**
Adjust the column width by means of the separator.
5 Navigation column of **TNCguide**

Symbols

Symbols in the Help application

Symbol	Function
	Show start page The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., **TNCguide**). If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function.
	Display tutorials
← →	Navigate between the last opened contents
	Display or hide search results **Further information:** "Search in TNCguide", Page 85

Symbols in the integrated product aid TNCguide

Symbol	Function
	Display documentation structure The structure consists of the content headings. The structure serves for main navigation within the documentation.
	Display documentation index The index consists of important keywords. The index serves as an alternative navigation within the documentation.
< >	Display previous or next page within the documentation
« »	Display or hide the navigation
	Copy NC examples to clipboard **Further information:** "Copying NC examples to clipboard", Page 85

2.5.1 Search in TNCguide

Using the search function, you can search for the entered search terms within the open documentation.

Use the search function as follows:

- Enter a character string

> The entry field is located in the title bar, to the left of the Home symbol that you use for navigating to the start page.
>
> The search starts automatically after you have entered e.g. a letter.
>
> If you wish to delete the entry, use the X symbol within the entry field.

- > The control opens the column containing the search results.
- > The control marks references also within open content pages.
- Select the reference
- > The control opens the selected content.
- > The control continues displaying the results of the last search.
- Select an alternative reference if necessary
- Enter a new character string if required

2.5.2 Copying NC examples to clipboard

Use the copy function to copy NC examples from the documentation to the NC editor.

To use the copy function:

- Navigate to the desired NC example
- Expand **Notes on using NC programs**
- Read and follow **Notes on using NC programs**

  Further information: "Notes on using NC programs", Page 81

- Copy NC example to clipboard

- > The button switches colors while copying.
- > The clipboard contains the entire content of the copied NC example.
- Insert the NC example into the NC program
- Adapt the inserted content according to the **Notes on using NC programs**
- Use the Simulation mode to test the NC program

  Further information: "Simulation Workspace", Page 1535

2.6 Contacting the editorial staff

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

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

tnc-userdoc@heidenhain.de

3

About the Product

3.1 The TNC7

Every HEIDENHAIN control supports you with dialog-guided programming and finely detailed simulation. The TNC7 additionally offers you graphical or form-based programming so that you can attain the desired results with speed and reliability.

Software options and optional hardware extensions can be used for flexibly increasing the range of functions and ease of use.

Functionality enhancements make it possible to go beyond milling and drilling in order to perform turning and grinding operations, for example,

Further information: "Technology-Specific NC Programming", Page 233

Operation is made easier, for example, by using touch probes, handwheels or a 3D mouse.

Further information: "Hardware", Page 102

Definitions

Abbreviation	Definition
TNC	**TNC** is derived from the acronym **CNC** (computerized numerical control). The **T** (tip or touch) stands for the capability of entering NC programs directly at the control or to program them graphically using gestures.
7	The product number indicates the control generation. The range of functions depends on the enabled software options.

3.1.1 Proper and intended use

The information about proper and intended use supports you in safely handling a product such as a machine tool.

The control is a machine component but not a complete machine. This User's Manual describes the use of the control. Before using the machine and the control, read the OEM documentation in order to inform yourself about the safety-relevent factors, the necessary safety equipment and the requirements for qualified personnel.

> HEIDENHAIN sells controls designed for milling and turning machines as well as for machining centers with up to 24 axes. If you as a user face a different constellation, then contact the owner immediately.

HEIDENHAIN contributes additionally to enhancing your safety and that of your products, notably by taking into consideration the customer feedback. This results, for example, in function adaptations of the controls and safety precautions in the information products.

> Contribute actively to increasing the safety by reporting any missing or misleading information.
>
> **Further information:** "Contacting the editorial staff", Page 85

3.1.2 Intended place of operation

In accordance with the DIN EN 50370-1 standard for electromagnetic compatibility (EMC), the control is approved for use in industrial environments.

Definitions

Guideline	Definition
DIN EN 50370-1:2006-02	This standard deals, among other things, with interference emissions and immunity to interference of machine tools.

3.2 Safety precautions

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

The following safety precautions refer exclusively to the control as an individual component but not to the specific complete product, i.e. the machine tool.

Refer to your machine manual.

Before using the machine and the control, read the OEM documentation in order to inform yourself about the safety-relevent factors, the necessary safety equipment and the requirements for qualified personnel.

The following overview contains only the generally valid safety precautions. Comply with the additional safey precautions provided in the following chapters. Some of this information depends on the specific configuraton.

For ensuring maximum safety, all safety precautions are repeated at the relevant places within the chapters.

DANGER

Caution: hazard to the user!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- Devices should be connected or removed only by authorized service technicians
- Only switch on the machine via a connected handwheel or a secured connection

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

DANGER

Caution: hazard to the user!

The **AUTOSTART** function automatically starts the machining operation. Open machines with unsecured work envelopes pose a huge danger for the machine operator.

- Use the **AUTOSTART** function exclusively on enclosed machines

WARNING

Caution: hazard to the user!

Manipulated data records or software can lead to an unexpected behavior of the machine. Malicious software (viruses, Trojans, malware, or worms) can cause changes to data records and software.

- Check any removable memory media for malicious software before using them
- Start the internal web browser only from within the sandbox

NOTICE

Danger of collision!

Failure to notice deviations between the actual axis positions and those expected by the control (saved at shutdown) can lead to undesirable and unexpected axis movements. There is risk of collision during the reference run of further axes and all subsequent movements!

- Check the axis positions
- Only confirm the pop-up window with **YES** if the axis positions match
- Despite confirmation, at first only move the axis carefully
- If there are discrepancies or you have any doubts, contact your machine manufacturer

NOTICE

Caution: Danger to the tool and workpiece!

A power failure during the machining operation can cause uncontrolled “coasting” or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

- Use a low feed rate
- Please keep in mind that the traverse range monitoring is not available for non-referenced axes

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision when referencing the axes.

- Pay attention to the information on the screen
- If necessary, move to a safe position before referencing the axes
- Watch out for possible collisions

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

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

- Before activating a preset, check whether all columns contain values.

NOTICE

Danger of collision!

NC programs that were created on older controls can lead to unexpected axis movements or error messages on current control models. Danger of collision during machining!

- Check the NC program or program section using the graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode

NOTICE

Caution: Data may be lost!

If you do not properly remove a connected USB device during a data transfer, then data may be damaged or deleted!

- Use the USB port only for transferring or backing up data do not use it for editing and executing NC programs
- Use the **Eject** soft key to remove a USB device when data the transfer is complete

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss regardless of the control's status!

- Always shut down the control
- Only operate the main switch after being prompted on the screen

NOTICE

Danger of collision!

If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions, e.g. transformations. This means that there is a risk of collision during subsequent traversing movements!

- Use **GOTO** only when programming and testing NC programs
- Only use **Block scan** when executing NC programs

3.3 Software

This User's Manual describes the functions for setting up the machine as well as for programming and and running your NC programs. These functions are available for a control featuring the full range of functions.

The actual range of functions depends, among other things, on the enabled software options.
Further information: "Software options", Page 94

The table shows the NC software numbers described in this User's Manual.

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.

NC software number	Product
817620-17	TNC7
817621-17	TNC7 E
817625-17	TNC7 programming station

Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions for the machine.

Check, on the basis of the machine tool manual, whether the machine manufacturer has adapted the functions of the control.

Definition

Abbreviation	Definition
E	The suffix E indicates the export version of the control. In this version, Advanced Function Set 2 (software option 9) is restricted to 4-axis interpolation.

3.3.1 Software options

Software options define the range of functions of the control. The optional functions are either machine- or application-specific. The software options give you the possibility of adapting the control to your individual needs.

You can check which software options are enabled on your machine.

Further information: "Viewing of software options", Page 2108

Overview and definitions

The **TNC7** features various software options, each of which can be enabled separately and even subsequently by the machine manufacturer. The following overview includes only those software options that are relevant for you as user.

> The option numbers indicated in the User's Manual show you that a function is not included in the standard range of available functions.
>
> The Technical Manual provides information about additional software options that are relevant for the machine manufacturer.

> Keep in mind that particular software options also require hardware extensions.
>
> **Further information:** "Hardware", Page 102

Software option	Definition and application
Additional Axis (options 0 to 7)	**Additional control loop** A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required e.g. for detachable and motor-driven tilting tables.
Advanced Function Set 1 (option 8)	**Advanced functions (set 1)** On machines with rotary axes this software option enables the machining of multiple workpiece sides in a single setup. The software option includes the following functions: ■ Tilting the working plane, e.g. with **PLANE SPATIAL** **Further information:** "PLANE SPATIAL", Page 1059 ■ Programming of contours on the unrolled surface of a cylinder (e.g., by using Cycle **27 CYLINDER SURFACE**) **Further information:** "Cycle 27 CYLINDER SURFACE (option 8)", Page 1267 ■ Programming the rotary axis feed rate in mm/min with **M116** **Further information:** "Interpreting the feed rate for rotary axes as mm/min with M116 (option 8)", Page 1333 ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.

Software option	Definition and application
Advanced Function Set 2 (option 9)	**Advanced functions (set 2)** On machines with rotary axes this software option enables the simultaneous 5-axis machining of workpieces. This software option includes the following functions: ■ **TCPM** (tool center point management): Automatic tracking of linear axes during rotary axis positioning **Further information:** "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104 ■ Running of NC programs with vectors, including optional 3D tool compensation **Further information:** "3D tool compensation (option 9)", Page 1126 ■ Manual moving of axes in the active tool coordinate system **T-CS** ■ Linear interpolation in more than 4 axes (max. 4 axes for an export version) The advanced functions (set 2) can be used to produce free-form surfaces.
HEIDENHAIN DNC (option 18)	**HEIDENHAIN DNC** This software option enables external Windows applications to access data in the control via the TCP/IP protocol. Potential fields of application are e.g.: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
Dynamic Collision Monitoring (option 40)	**Dynamic Collision Monitoring (DCM)** The machine manufacturer can use this software option to define machine components as collision objects. The control monitors the defined collision objects during all machine movements. The software option includes the following functions: ■ Automatic interruption of the program run whenever a collision is imminent ■ Warnings in the event of manual axis movements ■ Collision monitoring in Test Run mode With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or a machine downtime. **Further information:** "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
CAD Import (option 42)	**CAD Import** This software option is used to select positions and contours from CAD files and to transfer them into an NC program. With the CAD Import option you reduce the programming effort and prevent typical errors such as the incorrect entry of values. In addition, CAD Import contributes to paperless manufacturing. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1466

Software option	Definition and application
Global PGM Settings (option 44)	**Global Program Settings GPS** This software option can be used for superimposed coordinate transformations and handwheel movements during program run without adapting the NC program. With GPS you can adapt externally created NC programs to the machine and increase flexibility during program run. **Further information:** "Globale Programmeinstellungen GPS", Page
Adaptive Feed Control (option 45)	**Adaptive Feed Control AFC** This software option enables automatic feed control that depends on the current spindle load. The control increases the feed rate as the load decreases, and reduces the feed rate as the load increases. With AFC you can shorten the machining time without adapting the NC program, while preventing machine damage from overload at the same time. **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 1196
KinematicsOpt (option 48)	**KinematicsOpt** This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsOpt, the control can compensate for errors on rotary axes and thus increase accuracy during machining operations in the tilted working plane and during simultaneous machining operations. **Further information:** "Touch Probe Cycles: Automatic Kinematics Measurement", Page 1866
Turning (option 50)	**Mill-turning** This software option offers a comprehensive turning-specific package of functions for milling machines with rotary tables. The software option includes the following functions: ■ Turning-specific tools ■ Turning-specific cycles and contour elements such as undercuts ■ Automatic tool radius compensation Mill-turning enables mill-turning machining operations on only one machine, thus reducing e.g. the setup work considerably. **Further information:** "Turning (option 50)", Page 236
KinematicsComp (option 52)	**KinematicsComp** This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsComp the control can correct position and component errors in three dimensions. This means it can compensate for the errors of rotary and linear axes in three dimensions. Compared to KinematicsOpt (option 48), compensation is far more extensive. **Further information:** "Cycle 453 KINEMATICS GRID ", Page 1899

Software option	Definition and application
OPC UA NC Server 1 to **6** (options 56 to 61)	**OPC UA NC Server** These software options offer the OPC UA standardized interface for external access to data and functions of the control. Potential fields of application are e.g.: - Connection to higher-level ERP or MES systems - Capture of machine and operating data Each software option enables one client connection each. Several parallel connections require the use of multiple OPC UA NC servers. **Further information:** "OPC UA NC Server (options 56 to 61)", Page 2123
4 Additional Axes (option 77)	**4 additional control loops** **Further information:** "**Additional Axis** (options 0 to 7)", Page 94
8 Additional Axes (option 78)	**8 additional control loops** **Further information:** "**Additional Axis** (options 0 to 7)", Page 94
3D-ToolComp (option 92)	**3D-ToolComp** only in conjunction with Advanced Function Set 2 (option 9) With this software option, shape deviations on ball cutters and workpiece probes can be automatically compensated for using a compensation value table. 3D-ToolComp enables increasing the workpiece accuracy in conjunction with free-form surfaces, for example. **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1140
Extended Tool Management (option 93)	**Extended tool management** This software option extends tool management by the two tables **Tooling list** and **T usage order**. The tables show the following contents: - The **Tooling list** to be run shows the tool requirements of the NC program to be run or the pallet shows the tool requirements of the NC program **Further information:** "Tooling list (option 93)", Page 2033 - The **T usage order** shows the tool order of the NC program to be run or the pallet. **Further information:** "T usage order (option 93)", Page 2031 Extended Tool Management enables you to detect the tool requirements in time and thus prevent interruptions during program run.

Software option	Definition and application
Advanced Spindle Interpolation (option 96)	**Interpolating Spindle** This software option enables interpolation turning by causing the control to couple the tool spindle with the linear axes. This software option includes the following cycles: ■ Cycle **291 COUPLG.TURNG.INTERP.** for simple turning operations without contour subprograms **Further information:** "Cycle 291 COUPLG.TURNG.INTERP. (option 96)", Page 686 ■ Cycle **292 CONTOUR.TURNG.INTRP.** for finishing rotationally symmetrical contours **Further information:** "Cycle 292 CONTOUR.TURNG.INTRP. (option 96)", Page 694 The interpolating spindle also allows you to perform a turning operation on machines without a rotary table.
Spindle Synchronism (option 131)	**Spindle synchronism** This software option synchronizes two or more spindles and thus enables e.g. gear manufacturing via hobbing. This software option includes the following functions: ■ Spindle synchronism for special machining operations, e.g. polygonal turning ■ Cycle **880 GEAR HOBBING** only in conjunction with mill-turning (option 50) **Further information:** "Cycle 880 GEAR HOBBING (option 131)", Page 971
Remote Desktop Manager (option 133)	**Remote Desktop Manager** This software option is used to display and operate externally linked computer units. With Remote Desktop Manager you reduce the distances covered between several workplaces and as a result increase efficiency. **Further information:** "Remote Desktop Manager window (option 133)", Page 2137
Dynamic Collision Monitoring v2 (option 140)	**Dynamic Collision Monitoring (DCM) version 2** This software option includes the functions of software option 40 (Dynamic Collision Monitoring, DCM). In addition, this software option can be used for the collision monitoring of workpiece fixtures. **Further information:** "Integrating the fixtures into collision monitoring (option 140)", Page 1174
Cross Talk Compensation (option 141)	**Compensation of axis couplings CTC** Using this software option, the machine manufacturer can, for example, compensate for acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.
Position Adaptive Control (option 142)	**Position adaptive control PAC** Using this software option, the machine manufacturer can, for example, compensate for position-induced deviations at the tool and thus increase accuracy and dynamic performance.

Software option	Definition and application
Load Adaptive Control (option 143)	**Load adaptive control LAC** Using this software option, the machine manufacturer can, for example, compensate for load-induced deviations at the tool and thus increase accuracy and dynamic performance.
Motion Adaptive Control (option 144)	**Motion adaptive control MAC** Using this software option, the machine manufacturer can, for example, change speed-dependent machine settings and thus increase the dynamic performance.
Active Chatter Control (option 145)	**Active chatter control ACC** With this software option the chatter tendency of a machine used for heavy machining can be reduced. The control can use ACC to improve the surface quality of the workpiece, increase the tool life and reduce the machine load. Depending on the type of machine, the material-removal rate can be increased by more than 25 %. **Further information:** "Active Chatter Control (ACC, option 145)", Page 1204
Machine Vibration Control (option 146)	**Vibration damping for machines MVC** Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD **Active Vibration Damping** ■ FSC **Frequency Shaping Control**
CAD Model Optimizer (option 152)	**Optimization of CAD models** This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. **Further information:** "Generating STL files with 3D mesh (option 152)", Page 1472
Batch Process Manager (option 154)	**Batch Process Manager BPM** This software option makes it easy to plan and execute multiple production jobs. If pallet management and extended tool management (option 93) are extended or combined, BPM offers the following additional data, for example: ■ Machining time ■ Availability of necessary tools ■ Manual interventions to be made ■ Program test results of assigned NC programs **Further information:** "Job list workspace", Page 1938
Component Monitoring (option 155)	**Component monitoring** This software option enables the automatic monitoring of machine components configured by the machine manufacturer. Component monitoring assists the control in preventing machine damage due to overload by way of hazard warnings and error messages.

Software option	Definition and application
Grinding (option 156)	**Jig grinding** This software option offers a comprehensive grinding-specific package of functions for milling machines. The software option includes the following functions: ▪ Grinding-specific tools including dressing tools ▪ Cycles for reciprocating stroke and dressing Jig-turning enables complete machining operations on just one machine, thus reducing e.g. the setup work considerably. **Further information:** "Grinding operations (option 156)", Page 248
Gear Cutting (option 157)	**Gear manufacturing** This software option enables the manufacturing of cylindrical gears or helical gears of any angle. The software option includes the following cycles: ▪ Cycle **285 DEFINE GEAR** to define the gear geometry **Further information:** "Cycle 285 DEFINE GEAR (option 157)", Page 984 ▪ Cycle **286 GEAR HOBBING** **Further information:** "Cycle 286 GEAR HOBBING (option 157)", Page 986 ▪ Cycle **287 GEAR SKIVING** **Further information:** "Cycle 287 GEAR SKIVING option 157", Page 994 Gear manufacturing expands the scope of functionality of milling machines with rotary tables even without mill-turning (option 50).
Turning v2 (option 158)	**Mill-turning version 2** This software option includes all functions of Mill-Turning (software option 50). In addition, this software option offers the following advanced turning functions: ▪ Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** **Further information:** "Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option158) ", Page 887 ▪ Cycle **883 TURNING SIMULTANEOUS FINISHING** **Further information:** "Cycle 883 TURNING SIMULTANEOUS FINISHING (option 158)", Page 893 The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation.
Model Aided Setup (option 159)	**Graphically supported setup** This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions. The control supports you additionally by showing the clamping situation and possible touch points in the **Simulation** workspace by means of a 3D model.

Software option	Definition and application
Optimized Contour Milling (option 167)	**Optimized contour machining (OCM)** This software option enables the trochoidal milling of closed or open pockets and islands of any shape. During trochoidal milling, the full cutting edge is used under constant cutting conditions. The software option includes the following cycles: ■ Cycle **271 OCM CONTOUR DATA** ■ Cycle **272 OCM ROUGHING** ■ Cycle **273 OCM FINISHING FLOOR** and cycle **274 OCM FINISHING SIDE** ■ Cycle **277 OCM CHAMFERING** ■ In addition, the control offers **OCM STANDARD FIGURES** for frequently needed contours With OCM you can shorten the machining time while reducing tool wear at the same time. **Further information:** "OCM cycles", Page 656
Process Monitoring (option 168)	**Process monitoring** Reference-based monitoring of the machining process The control uses this software option to monitor defined machining sections during program run. The control compares changes with regard to the tool spindle or the tool with the values of a reference machining operation. **Further information:** "Arbeitsbereich Prozessüberwachung (Option #168)", Page

3.3.2 Information on licensing and use

Open-source software

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

To get to the licensing terms on the control:

- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Operating system** tab
- Double-tap or double-click **About HeROS**
- > The control opens the **HEROS Licence Viewer** window.

OPC UA

The control software contains binary libraries, to which the terms of use agreed between HEIDENHAIN and Softing Industrial Automation GmbH additionally and preferentially apply.

OPC UA NC Server (options 56 to 61) and HEIDENHAIN DNC (option 18) can be used to influence the behavior of the control. Before using these interfaces for productive purposes, system tests must be performed to exclude the occurrence of any malfunctions or performance failures of the control. The manufacturer of the software product that uses these communication interfaces is responsible for performing these tests.

Further information: "OPC UA NC Server (options 56 to 61)", Page 2123

3.4 Hardware

This User's Manual describes functions for setting up and operating the machine. These functions primarily depend on the installed software.

Further information: "Software", Page 93

The actual range of functions also depends on hardware enhancements and the enabled software options.

3.4.1 Monitor

BF 360

The TNC7 is delivered with a 24-inch touchscreen.

The control is operated by means of touchscreen gestures and with the operating elements of the keyboard unit.

Further information: "Common gestures for the touchscreen", Page 116

Further information: "Operating elements of the keyboard unit", Page 116

Operation and cleaning

> **Avoiding electrostatic discharge when operating touchscreens**
>
> Touchscreens are based on a capacitive working principle, i.e. they are sensitive to electrostatic charges generated by the operators.
>
> Users can discharge static electricity from their body by touching grounded metal objects. This problem can be avoided by wearing ESD clothing.

Capacitative sensors detect a contact as soon as a person's finger touches the touchscreen. Touchscreens can even be operated with dirty hands, as long as the touch sensors are able to detect the skin resistance. While small amounts of liquid will not cause a fault, larger quantities of liquid will cause erroneous input.

> Use work gloves to prevent the device from becoming dirty. The rubber material of special touchscreen work gloves contains metal ions that transfer the skin resistance to the display.

In order to maintain the functionality of the touchscreen, use the following cleaners only:

- Glass cleaner
- Foaming screen cleaners
- Mild detergents

> Do not apply the cleaner directly to screen, but slightly dampen a suitable cleaning cloth with it.

Switch off the control before cleaning the touchscreen. As an alternative, you can use the touchscreen cleaning mode.

Further information: "Settings Application", Page 2099

> Never use the following cleaners or cleaning aids, in order to avoid damage to the touchscreen:
>
> - Aggressive solvents
> - Abrasives
> - Compressed air
> - Steam cleaners

3.4.2 Keyboard unit

TE 360 with standard potentiometer layout

TE 360 with alternative potentiometer layout

TE 361

The TNC7 is delivered with various keyboard units.

The control is operated by means of touchscreen gestures and with the operating elements of the keyboard unit.

Further information: "Common gestures for the touchscreen", Page 116

Further information: "Operating elements of the keyboard unit", Page 116

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

> Use operating gloves to prevent the device from becoming dirty.

In order to maintain the functionality of the keyboard, use only cleaners stated to contain anionic or nonionic surfactants.

> Do not apply the cleaner directly to the keyboard unit. Slightly dampen a suitable cleaning cloth with the cleaner.

Switch the control off before cleaning the keyboard unit.

> Never use the following cleaners or cleaning aids, in order to avoid damage to the keyboard unit:
> - Aggressive solvents
> - Abrasives
> - Compressed air
> - Steam blasters

> The trackball does not require periodic maintenance. Cleaning is required only if the trackball stops functioning.

If a trackball is embedded in the keyboard, clean the trackball as follows:

- ▶ 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 an isopropyl alcohol cleaner to a lint-free and clean cloth

  > Please observe the information for the cleaner.

- ▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

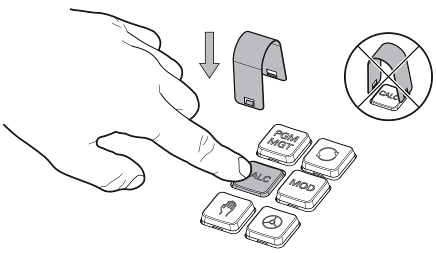
Exchanging keycaps

If you need replacements for the keycaps of the keyboard unit, contact HEIDENHAIN or the machine manufacturer.

Further information: "Keycaps for keyboard units and machine operating panels", Page 2321

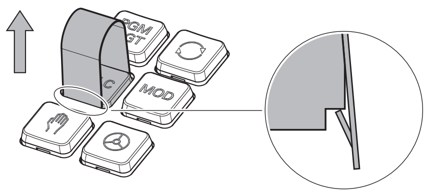
⚙	The IP54 protection rating cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:

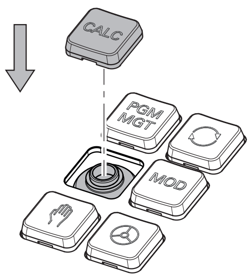


- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

ℹ	Pressing the key will make it easier to apply the keycap puller.



- Pull off the keycap



- Place the keycap onto the seal and push it down

ℹ	The seal must not be damaged; otherwise the IP54 protection rating cannot be guaranteed.

- Verify proper seating and correct functioning

3.4.3 Hardware enhancements

The hardware enhancements give you the possibility of adapting the machine tool to your individual needs.

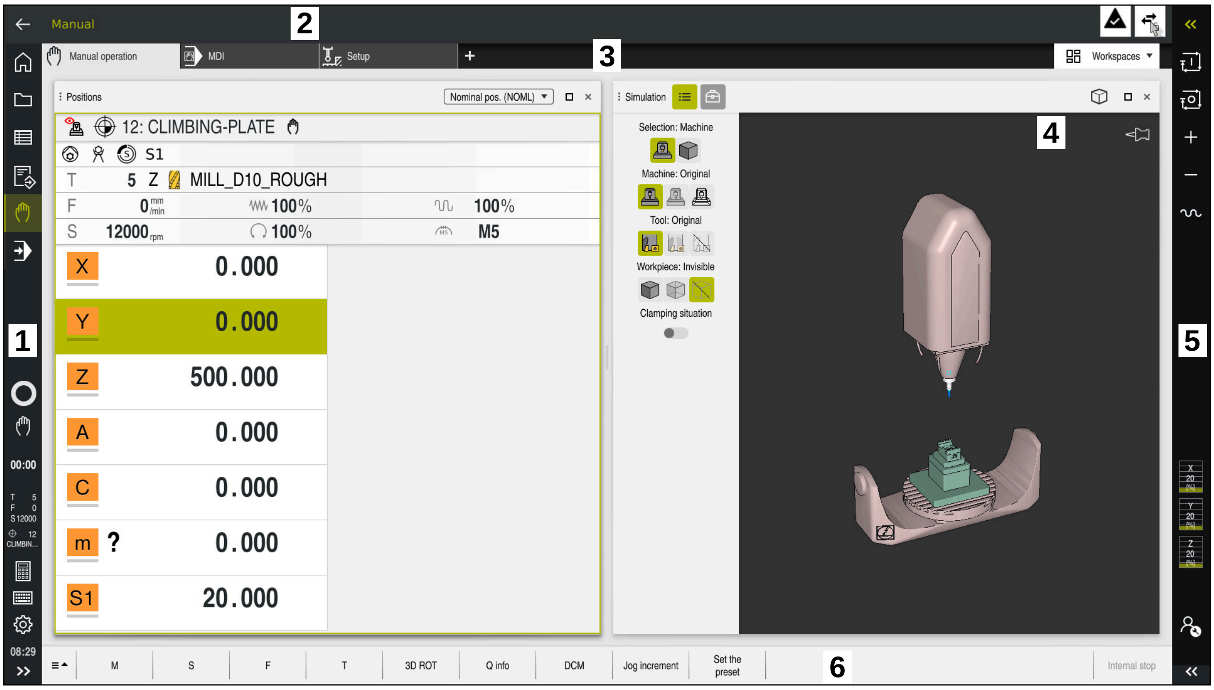
The **TNC7** features various hardware enhancements, each of which can be added separately and even subsequently by the machine manufacturer. The following overview includes only those enhancements that are relevant for you.

> Keep in mind that particular hardware enhancements require additional software options.
>
> **Further information:** "Software options", Page 94

Hardware enhancements	Definition and application
Electronic handwheels	You use this enhancement for exact manual positioning of machine axes. The wireless portable variants improve ergonomics and increase versatility. The handwheels have the following differing features: ■ Portable or installed in the machine operating panel ■ With or without display ■ With or without functional safety Electronic handwheels, for example, greatly simplify workpiece setup. **Further information:** "Electronic Handwheel", Page 2069
Workpiece touch probes	The control uses this enhancement for automatic and precise detection of workpiece positions and misalignments. The workpiece touch probes have the following differing features: ■ With radio or infrared transmission ■ With or without cable Workpiece touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions during program run. **Further information:** "Touch Probe Functions in the Manual Operating Mode", Page 1557
Tool touch probes	The control uses this enhancement for automatic and precise calibration of tools directly in the machine. Tool touch probes have the following differing features: ■ Contact-free or tactile measurement ■ With radio or infrared transmission ■ With or without cable Tool touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions and breakage control during program run. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906

Hardware enhancements	Definition and application
Vision systems	Use this enhancement to inspect the tools used. With the VT 121 vision system, you can visually inspect the cutting edges during program run without removing the tool. The vision systems help to avoid damage during program run, thus preventing unnecessary costs. **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
Additional operating stations	This enhancement adds a second screen, to facilitate operation of the control. The additional ITC (industrial thin client) operating stations are differentiated by their intended use: ■ The ITC 755 is a compact, additional operating station that mirrors the control's main screen, making it possible to operate the control. ■ The ITC 860 is an auxiliary screen that increases the area of the main screen. This allows multiple applications to be viewed simultaneously. By adding a keyboard unit, the ITC 860 can be used as a full-fledged additional operating station. The additional operating stations increase operator comfort, especially on large machining centers.
Industrial PC	You use this enhancement to install and run Windows-based applications. With Remote Desktop Manager (option 133), you can display the applications on the control's screen. **Further information:** "Remote Desktop Manager window (option 133)", Page 2137 The industrial PC is a secure and powerful alternative to external PCs.

3.5 Areas of the control's user interface



The control's user interface in the **Manual operation** application

The control's user interface shows the following areas:

1 TNC bar

- Back

  Use this function to go backwards in the application history since booting the control.
- Operating modes

  Further information: "Overview of operating modes", Page 110
- Status overview

  Further information: "Status overview on the TNC bar", Page 169
- Calculator

  Further information: "Calculator", Page 1527
- Screen keyboard

  Further information: "Virtual keyboard of the control bar", Page 1508
- Settings

  The Settings menu enables you to change the control interface:
  - **Left-hand mode**

    The control swaps the positions of the TNC bar and the machine manufacturer bar.
  - **Dark Mode**
  - **Font size**
- Date and time

2 Information bar

- Active operating mode
- Message menu

  Further information: "Message menu on the information bar", Page 1532
- Symbols

3 Application bar
- Tabs of opened applications

  The maximum number of simultaneously opened applications is limited to ten tabs. If you try to open an eleventh tab, the control shows a message.
- Selection menu for workspaces

  With the selection menu you define which workspaces are open in the active application.

4 Workspaces

Further information: "Workspaces", Page 112

5 Machine manufacturer bar

The machine manufacturer configures the machine manufacturer bar.

6 Function bar
- Selection menu for buttons

  With the selection menu you define which buttons the control displays in the function bar.
- Button

  With the buttons you activate individual functions of the control.

3.6 Overview of operating modes

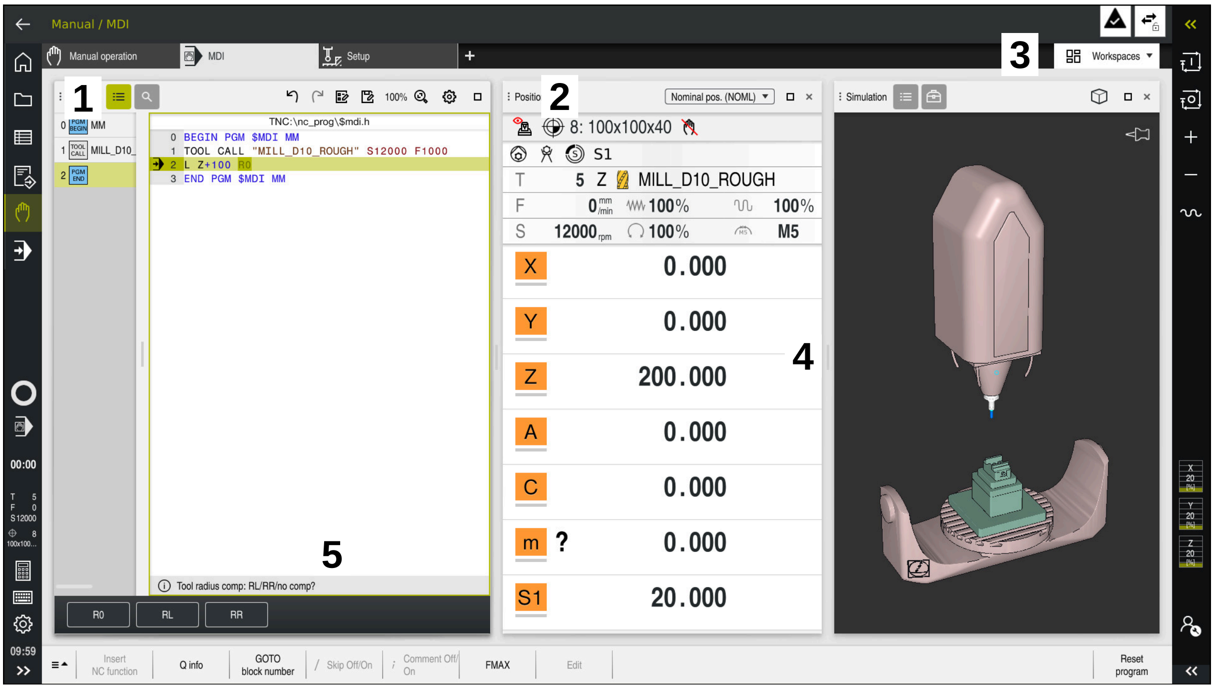
The control provides the following operating modes:

Symbols	Operating modes	Further information
	The **Home** operating mode contains the following applications:	
	■ **Start/Login** application During the startup process, the control is in the **Start/Login** application.	
	■ **Settings** application	Page 2099
	■ **Help** application	Page 1506
	■ Applications for machine parameters	Page 2152
	In the **Files** operating mode the control displays drives, folders and files. You can, for example, create or delete folders or files and can also connect drives.	Page 1144
	In the **Tables** operating mode you can open various tables and edit them as necessary.	Page 1980
	In the **Editor** operating mode you can do the following: ■ Create, edit and simulate NC programs ■ Create and edit contours ■ Create and edit pallet tables	Page 216

Symbols	Operating modes	Further information
	The **Manual** operating mode contains the following applications:	
	■ **Manual operation** application	Page 202
	■ **MDI** application	Page 1933
	■ **Setup** application	Page 1557
	■ **Move to ref. point** application	Page 197
	In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either one block at a time or in full sequence. You also execute pallet tables in this operating mode.	Page 1954
	In the **Retract** application you can move the tool away from the workpiece, for example after a power failure.	Page 1976
	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode. Refer to your machine manual.	Page 2087
	In the **Machine** operating mode, the machine manufacturer defines his own functions, such as diagnostic functions for spindle and axes, or applications. Refer to your machine manual.	

3.7 Workspaces

3.7.1 Operating elements within the workspaces



The control in the **MDI** application with three open workspaces

The control displays the following operating elements:

1 Gripper
Use the gripper in the title bar to change positions of the workspaces. You can also align two workspaces vertically above each other.

2 Title bar
In the title bar the control shows the title of the workspace, and different symbols or settings, depending on the workspace.

3 Selection menu for workspaces
Use the selection menu for workspaces in the application bar to open individual workspaces. The available workspaces depend on the active application.

4 Separator
You use the separator between two workspaces to change the scaling of the workspaces.

5 Action bar
In the action bar the control shows selection possibilities for the current dialog; for example, an NC function.

3.7.2 Symbols within the workspaces

If more than one workspace is open, the title bar contains the following symbols:

Icon	Function
☐	Maximize workspace
🗗	Reduce workspace
✕	Close workspace

If you maximize a workspace, the control shows the workspace over the application's entire area. If you reduce the workspace, then all other workspaces return to their previous position.

3.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function In the **Probing function** workspace you set presets on the workpiece and determine and compensate for workpiece misalignment and rotations. You can also calibrate the touch probe, measure tools, and set up fixtures.	Page 1557
Job list In the **Job list** workspace, you edit and execute pallet tables.	Page 1938
Open File In the **Open File** workspace you can select and create files, for example.	Page 1153
Document In the **Document** workspace, you can open files in order to view them, such as a technical drawing.	Page 1154
Form for tables In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.	Page 1989
Form for pallets In the **Form** workspace, the control shows the contents of the pallet table for the selected row.	Page 1946
Retract In the **Retract** workspace you disengage the tool after a power interruption.	Page 1976
GS (option 44) In the **GS** workspace you define selected transformations and settings without modifying the NC program.	Page 1217
Desktop menu In the **Desktop menu** workspace the control displays selected control and HEROS functions.	Page 125

Workspace	Further information
Help In the **Help** workspace the control displays a help graphic for the current syntax element of an NC function or the **TNCguide** integrated product aid.	Page 1506
Contour In the **Contour** workspace you use lines and arcs to draw a 2D sketch and then generate a Klartext contour from it. You can also import program sections with contours from an NC program to the **Contour** workspace for graphical editing.	Page 1437
List In the **List** workspace the control shows the machine parameter structure; you might be able to edit some of the parameters.	Page 2153
Positions In the **Positions** workspace the control displays information about the status of various functions of the control and about current axis positions.	Page 163
Program The control displays the NC program in the **Program** workspace.	Page 217
RDP (option 133) If the machine manufacturer has defined an embedded workspace, you can see and operate the screen of an external computer on the control. The machine manufacturer can change the name of the workspace. Refer to your machine manual.	Page 2087
Quick selection In the **Quick selection** workspace, you can create files or open existing ones regardless of the active operating mode.	Page 1154
Simulation In the **Simulation** workspace the control shows the simulated or actual movements, depending on the operating mode.	Page 1535
Simulation status In the **Simulation status** workspace the control shows data based on the simulation of the NC program.	Page 186
Start/Login In the **Start/Login** workspace the control shows the steps that are performed while booting.	Page 128
Status In the **Status** workspace the control shows the status and values of individual functions.	Page 171
Table In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.	Page 1982
Table for machine parameters In the **Table** workspace the control shows the machine parameters; you might be able to edit some of them.	Page 2153

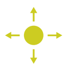
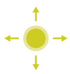
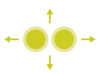
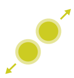
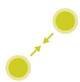
Workspace	Further information
Keyboard In the **Keyboard** workspace you can enter NC functions, letters and numbers, and also navigate.	Page 1508
Overview In the **Overview** workspace the control displays the information on the status of individual functional safety (FS) aspects.	Page 2094
Monitoring In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.	Page 1238

3.8 Operating elements

3.8.1 Common gestures for the touchscreen

The screen of the control is multi-touch capable. That means the control can distinguish various gestures, even with two or more fingers at once.

You can use the following gestures:

Symbol	Gesture	Meaning
	Tap	A brief touch by a finger on the screen
	Double tap	Two brief touches on the screen
	Long press	Continuous contact of finger tip on the screen If you do not stop holding, the control will automatically cancel the holding gesture after approximately ten seconds. Permanent actuation is thus not possible.
	Swipe	Flowing motion over the screen
	Drag	A combination of long-press and then swipe, moving a finger over the screen when the starting point is clearly defined
	Two-finger drag	A combination of long-press and then swipe, moving two fingers in parallel over the screen when the starting point is clearly defined
	Spread	Two fingers long-press and move away from each other
	Pinch	Two fingers move toward each other

3.8.2 Operating elements of the keyboard unit

Application

You operate the TNC7 primarily through the touchscreen, meaning with gestures.

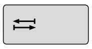
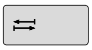
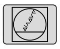
Further information: "Common gestures for the touchscreen", Page 116

In addition, the control's keyboard unit offers keys and other elements for alternative operating sequences.

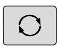
Description of function

The tables below describe the keyboard unit's operating elements.

Keycaps for alphabetic keyboard

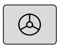
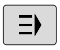
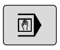
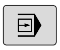
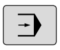
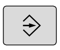
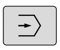
Key	Function
A B C	Enter texts (e.g., file names)
SHIFT + Q	**Uppercase Q** If an NC program is open, in the **Editor** operating mode for entering a Q parameter formula; in the **Manual** operating mode for opening the **Q parameter list** window **Further information:** "Q parameter list window", Page 1366
ESC	Close windows and context menus
 **SHIFT +** 	Select the next element, e.g. input field, button, selection option Select the previous element
PRT SC	Create screenshot
	**Left DIADUR key** Open the
	Open the context menu in **Klartext programming** or in the text editor

Keycaps for operating aids

Key	Function
PGM MGT	Open the **Open File** workspace in the **Editor** and **Program Run** operating modes **Further information:** "Open File workspace", Page 1153
	Select the first right-aligned button in the function bar
ERR	Open and close the message menu **Further information:** "Message menu on the information bar", Page 1532
CALC	Open and close the calculator **Further information:** "Calculator", Page 1527
MOD	Open the **Settings** application **Further information:** "Settings Application", Page 2099
HELP	Open the online help **Further information:** "User's Manual as integrated product aid: TNCguide", Page 82

Operating modes

> On the TNC7 the operating modes of the control are allocated differently than on the TNC 640. For reasons of compatibility and to facilitate ease of operation, the keys on the keyboard unit remain the same. Keep in mind that particular keys no longer activate a change of operating modes but, for example, instead activate a toggle switch.

Key	Function
	Opening the **Manual operation** application in the **Manual** operating mode **Further information:** "Manual operation application", Page 202
	Activating and deactivating the electronic handwheel in the **Manual** operating mode **Further information:** "Electronic Handwheel", Page 2069
	Opening the **Tool Management** tab in the **Tables** operating mode **Further information:** "Tool management ", Page 297
	Opening the **MDI** application in the **Manual** operating mode **Further information:** "The MDI Application ", Page 1933
	Opening the **Program Run** operating mode in **Single Block** mode **Further information:** "Program Run operating mode", Page 1954
	Opening the **Program Run** operating mode **Further information:** "Program Run operating mode", Page 1954
	Opening the **Editor** operating mode **Further information:** "Editor operating mode", Page 216
	While the NC program is running, opening of the **Simulation** workspace in the **Editor** operating mode **Further information:** "Simulation Workspace", Page 1535

Keycaps for NC dialog

The following functions are valid for the **Editor** operating mode and the **MDI** application.

Key	Function
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function **Further information:** "Fundamentals of approach and departure functions", Page 355
FK	Open the **Contour** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode **Further information:** "Graphical Programming", Page 1437
CHF	Program a chamfer **Further information:** "Chamfer CHF", Page 328
L	Program a straight line segment **Further information:** "Straight line L", Page 326
CR	Program a circular arc with radius entry **Further information:** "Circular path CR", Page 336
RND	Program a rounding arc **Further information:** "Rounding RND", Page 330
CT	Program a circular arc with tangential connection to the preceding contour element **Further information:** "Circular path CT", Page 339
CC	Program a circle center or pole **Further information:** "Circle center point CC", Page 332
C	Program a circular arc with reference to the circle center **Further information:** "Circular path C ", Page 334
TOUCH PROBE	In the **Insert NC function** window, open the **Setup** folder in order to select a touch probe cycle **Further information:** "Programmable Touch Probe Cycles", Page 1589
CYCL DEF	In the **Insert NC function** window, open the **Fixed cycles** folder in order to select a cycle **Further information:** "Defining cycles", Page 478
CYCL CALL	In the **Insert NC function** window, open the **Cycle call** folder in order to select a machining cycle **Further information:** "Calling cycles", Page 481
LBL SET	Program a jump label **Further information:** "Defining a label with LBL SET", Page 384
LBL CALL	Program a subprogram or a program section repeat **Further information:** "Calling a label with CALL LBL", Page 385

Key	Function
STOP	Program an intentional stop **Further information:** "Programming the STOP function", Page 1318
TOOL DEF	Pre-select a tool in the NC program **Further information:** "Tool pre-selection by TOOL DEF", Page 311
TOOL CALL	Call the tool data in the NC program **Further information:** "Tool call by TOOL CALL", Page 304
SPEC FCT	In the **Insert NC function** window, open the **Special functions** folder (e.g., for later programming of a workpiece blank)
PGM CALL	In the **Insert NC function** window, open the **Selection** folder (e.g., to call an external NC program)

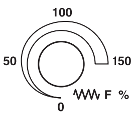
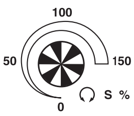
Keycaps for axis input and value input

Key	Function
X ... V	Select axes in the **Manual** operating mode, or enter them in the **Editor** operating mode
0 ... 9	Enter numbers (e.g., coordinate values)
.	Insert a decimal separator during entry
-/+	Invert algebraic sign of entered value
	Delete values during entry
	Open position display of the status overview to copy axis values **Further information:** "Status overview on the TNC bar", Page 169 In **Editor** operating mode and the **MDI** application, program a straight line **L** with the actual positions of all axes
Q	In the **Editor** operating mode, within the **Insert NC function** window, open the **FN** folder
CE	Clear entries or delete messages
DEL	Delete NC block or cancel a dialog during programming
NO ENT	Skip or remove optional syntax elements during programming
ENT	Confirm entries and continue dialogs
END	Conclude entry, e.g. finish an NC block
P	Switch between entry of polar and Cartesian coordinates
I	Switch between entry of incremental and absolute coordinates

Keycaps for navigation

Key	Function
↑ . . . →	Position the cursor
GOTO ☐	■ Position the cursor by using the block number of an NC block ■ Open the selection menu while editing
HOME	Jump to first line of an NC program or first column of a table
END	Jump to last line of an NC program or last column of a table
PG UP	Go one page up in an NC program or table
PG DN	Go one page down in an NC program or table
	Mark the active application in order to navigate between applications
 	Navigate between areas of an application

Potentiometers

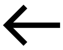
Potentiometer	Function
	Increase or reduce the feed rate **Further information:** "Feed rate F", Page 310
	Increase or reduce the spindle speed **Further information:** "Spindle speed S", Page 309

3.8.3 Icons on the control's user interface

Overview of icons not specific to any operating mode

This overview describes icons that are used in more than one operating mode or that are available regardless of operating mode.

Icons that are specific to individual workspaces are described there.

Icon or shortcut	Function
	Back
	Select the **Home** operating mode
	Select the **Files** operating mode
	Select the **Tables** operating mode
	Select the **Editor** operating mode
	Select the **Manual** operating mode
	Select the **Program Run** operating mode
	Select the **Machine** operating mode
	Open and close the calculator
	Open and close the virtual keyboard
	Open and close the settings
>>	■ White: Expand control bar or machine manufacturer bar ■ Green: Collapse control bar or machine manufacturer bar or go back ■ Gray: Confirm message
	Add
	Open file
	Close
	Maximize workspace
	Reduce workspace
	Change position of workspaces or windows
	Resize windows
	■ Black: Add to favorites ■ Yellow: Remove from favorites

Icon or shortcut	Function
CTRL+S	Save
	Save as
CTRL+F	Find
CTRL+C	Copy
CTRL+V	Paste
CTRL+Z	Undo an action
CTRL+Y	Redo an action
	Open selection menu
	Open message menu

3.8.4 Desktop menu workspace

Application

In the **Desktop menu** workspace the control displays selected control and HEROS functions.

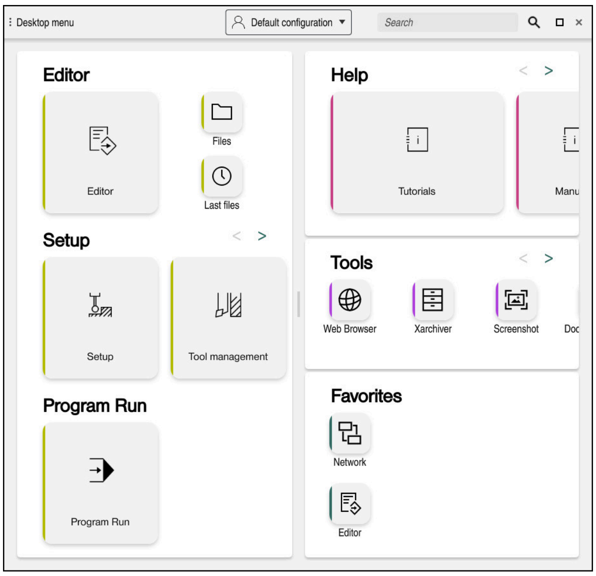
Description of function

The title bar of the **Desktop menu** workspace includes the following functions:

- **Active configuration** selection menu
  Using the selection menu, you can activate a configuration of the control interface.
  Further information: "Configuring the control's user interface", Page 2157
- Full-text search
  Search for functions in the workspace with the full-text search.
  Further information: "Adding and removing favorites", Page 126

The **Desktop menu** workspace contains the following areas:

- **Control**
  In this area you can open operating modes or applications.
  Further information: "Overview of operating modes", Page 110
  Further information: "Overview of workspaces", Page 113
- **Tools**
  In this area you can open some tools from the HEROS operating system.
  Further information: "HEROS Operating System", Page 2183
- **Help**
  In this area you can open training videos or **TNCguide**.
- **Favorites**
  In this area you will find the favorites that you have chosen.
  Further information: "Adding and removing favorites", Page 126



Desktop menu workspace

The **Desktop menu** workspace is available in the **Start/Login** application.

Showing or hiding an area

To show or hide an area in the **Desktop menu** workspace:

- ▶ Hold or right-click anywhere within the workspace
- > The control displays a plus sign or minus sign within each area.
- ▶ Select a plus sign
- > The controls shows that area.

> Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

To add favorites in the **Desktop menu** workspace:

- ▶ Use the full-text search
- ▶ Hold or right-click the function's icon
- > The control displays the icon for **adding favorites**.

- ▶ Select **Add favorite**
- > The control adds the function to the **Favorites** area.

Removing favorites

To remove favorites from the **Desktop menu** workspace:

- ▶ Hold or right-click the function's icon
- > The control displays the icon for **removing favorites**.

- ▶ Select **Remove favorite**
- > The control removes the function from the **Favorites** area.

4

First Steps

4.1 Chapter overview

This chapter uses an example workpiece to explain how to operate the control: from switching the machine on to the finished workpiece.

The chapter covers the following topics:

- Switching the machine on
- Programming and simulating a workpiece
- Setting up tools
- Setting up the workpiece
- Machining the workpiece
- Switching the machine off

4.2 Switching the machine and the control on

Start/Login workspace

⚠ 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 the machine on:

- Switch the power supply of the control and of the machine on
- > The control is in start-up mode and shows the progress in the **Start/Login** workspace.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace.

OK

- Press **OK**
- > The control compiles the PLC program.

I

- Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit.
- > If the machine is equipped with absolute linear and angle encoders, the control is now ready for operation.
- > If the machine is equipped with incremental linear and angle encoders, the control opens the **Move to ref. point** application.

  Further information: "Referencing workspace", Page 197

- Press the **NC Start** key
- > The control moves to all necessary reference points.
- > The control is ready for operation and the **Manual operation** application is open.

  Further information: "Manual operation application", Page 202

More detailed information

- Switching on and off

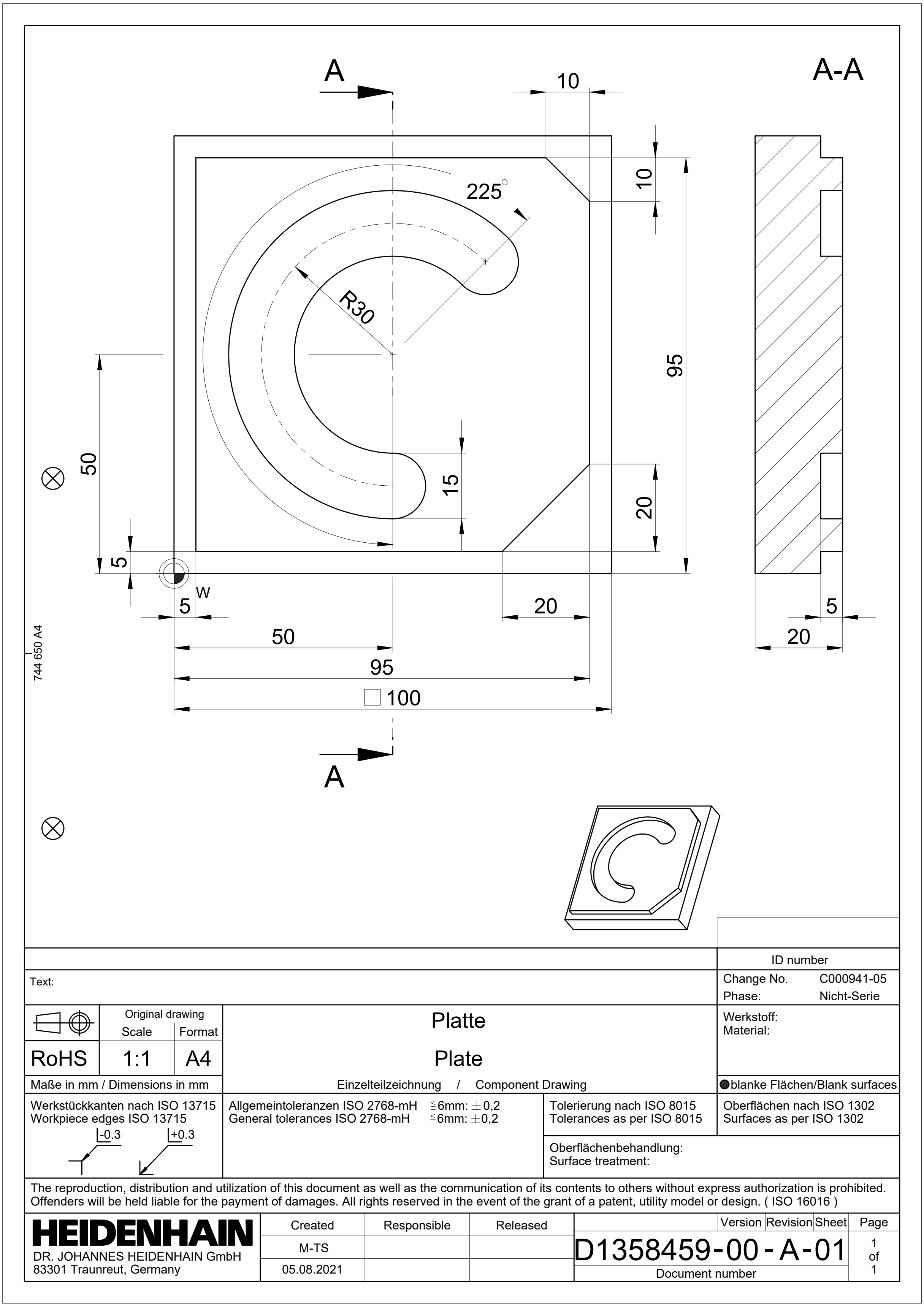
  Further information: "Powering On and Off", Page 193
- Position encoders

  Further information: "Position encoders and reference marks", Page 209
- Axis reference run

  Further information: "Referencing workspace", Page 197

4.3 Programming and simulating a workpiece

4.3.1 Example task 1338459



4.3.2 Selecting the Editor operating mode

NC programs are always programmed in the **Editor** operating mode.

Requirement

- It must be possible to select the icon of the operating mode

  In order to be able to select the **Editor** operating mode, the control must have already progressed enough during booting that the operating mode icon is no longer dimmed.

Selecting the Editor operating mode

To select the **Editor** operating mode:

- ▶ Select the **Editor** operating mode
- > The control displays the **Editor** operating mode and the most recently opened NC program.

More detailed information

- Operating mode: **Editor**

  Further information: "Editor operating mode", Page 216

4.3.3 Configuring the control's user interface for programming

The **Editor** operating mode gives you several possibilities for writing an NC program.

> The first steps describe the procedure when you are in the **Klartext programming** mode and the **Form** column is open.

Opening the Form column

You can open the **Form** column only if an NC program is open.

To open the **Form** column:

- ▶ Select **Form**
- > The control opens the **Form** column

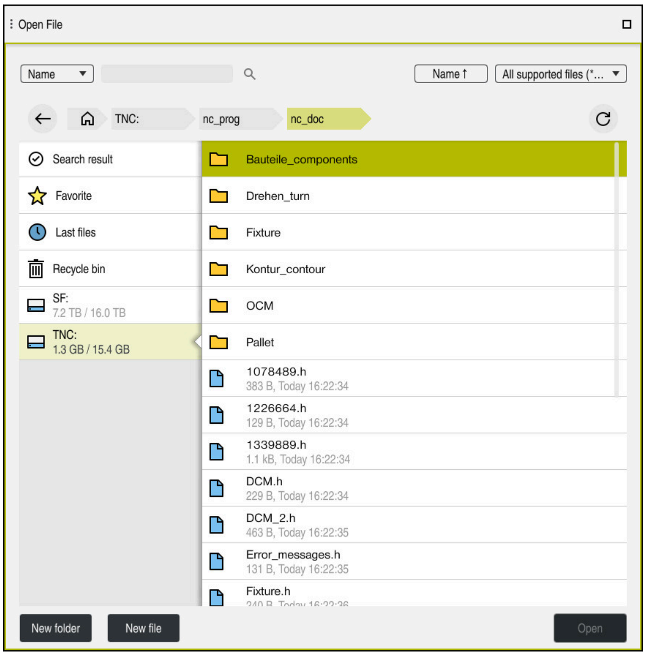
More detailed information

- Editing an NC program

  Further information: "Editing NC programs", Page 228
- Column: **Form**

  Further information: "Form column in the Program workspace", Page 227

4.3.4 Creating a new NC program



Open File workspace in the **Editor** operating mode

To create an NC program in the **Editor** operating mode:

- Select **Add**
- The control displays the **Quick selection** and **Open File** workspaces.
- Select the desired drive in the **Open File** workspace
- Select a folder
- New file: Select **New file**
- Enter a file name (e.g., 1338459.h)
- ENT: Confirm with the **ENT** key
- Open: Select **Open**
- The control opens a new NC program and the **Insert NC function** window for definition of the workpiece blank.

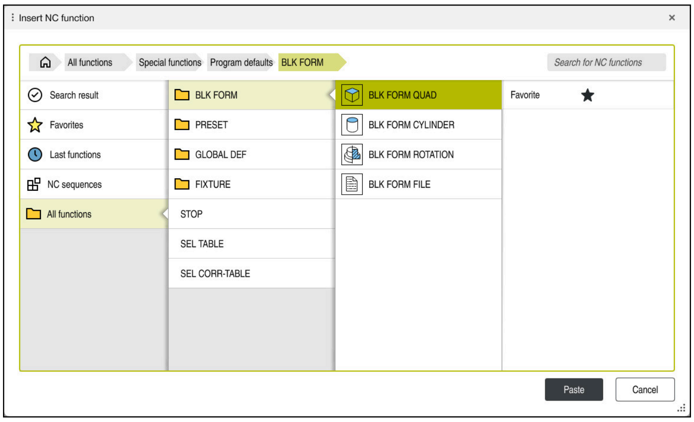
More detailed information

- Workspace: **Open File**
  Further information: "Open File workspace", Page 1153
- Operating mode: **Editor**
  Further information: "Editor operating mode", Page 216

4.3.5 Defining the workpiece blank

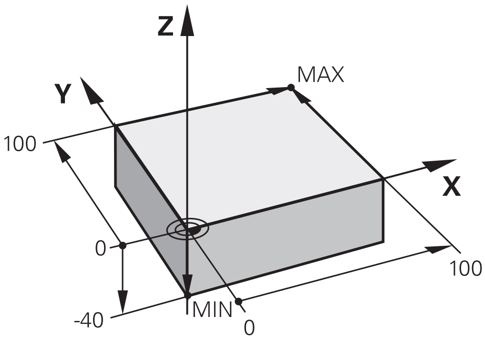
For the NC program you can define a workpiece blank that the control then uses for the simulation. When you create an NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

> If you close the window without selecting a workpiece blank, you can use the **Insert NC function** button to select the definition of the workpiece blank.



Insert NC function window for defining the workpiece blank

Defining a cuboid workpiece blank



Cuboid workpiece blank with minimum point and maximum point

You define a cuboid through a diagonal in space by entering the minimum point and maximum point relative to the active workpiece preset.

> You can confirm the entries as follows:
> - **ENT** key
> - Right arrow key
> - Click or tap the next syntax element

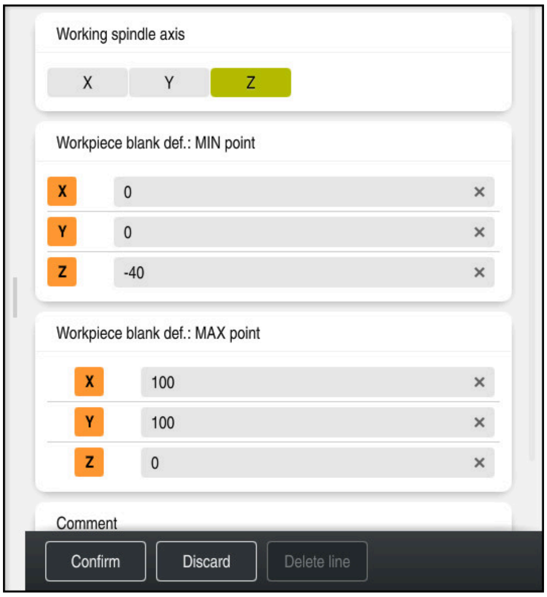
To define a cuboid workpiece blank:

- Select **BLK FORM QUAD**

- Select **Paste**
- The control inserts the NC block for definition of the workpiece blank.

- Open the **Form** column
- Select the tool axis (e.g., **Z**)
- Confirm your input
- Enter the smallest X coordinate (e.g., **0**)
- Confirm your input
- Enter the smallest Y coordinate (e.g., **0**)
- Confirm your input
- Enter the smallest Z coordinate (e.g., **-40**)
- Confirm your input
- Enter the largest X coordinate (e.g., **100**)
- Confirm your input
- Enter the largest Y coordinate (e.g., **100**)
- Confirm your input
- Enter the largest Z coordinate (e.g., **0**)
- Confirm your input

- Select **Confirm**
- The control concludes the NC block.



Form column with the defined columns

```
0 BEGIN PGM  1339889 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  1339889 MM
```

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.

More detailed information

- Inserting the workpiece blank

  Further information: "Defining a workpiece blank with BLK FORM", Page 258
- Reference points in the machine

  Further information: "Presets in the machine", Page 210

4.3.6 Structure of an NC program

Using a uniform structure for an NC program offers the following advantages:

- Improved overview
- Quicker programming
- Fewer sources of error

Recommended structure for a contouring program

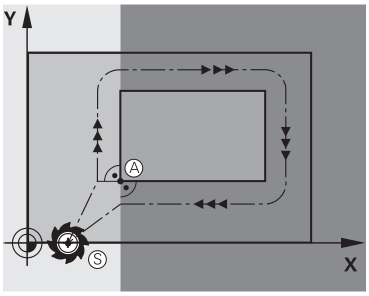
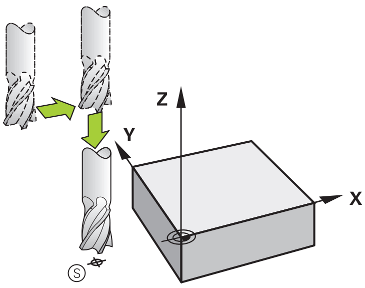
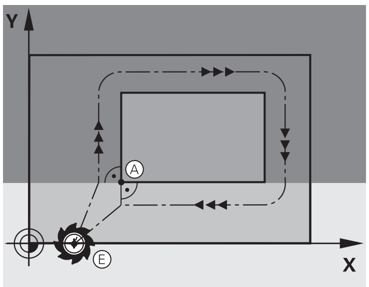
> The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks.

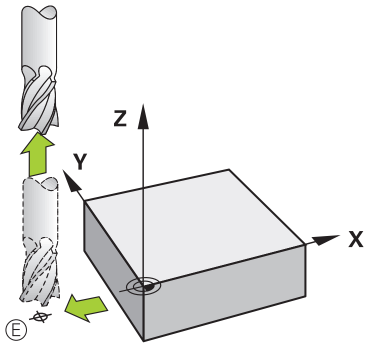
1 **BEGIN PGM** with selection of the unit of measure
2 Define the workpiece blank
3 Call the tool, with the tool axis and the technological data
4 Move the tool to a safe position, and switch the spindle on
5 Pre-position the tool in the working plane, near the first contour point
6 Pre-position the tool in the tool axis, turn coolant on if necessary
7 Approach the contour, activate tool radius compensation if necessary
8 Machine the contour
9 Depart from the contour, turn coolant off
10 Move the tool to a safe position
11 Conclude the NC program
12 **END PGM**

4.3.7 Contour approach and departure

When you program a contour, you need a starting point and end point outside the contour.

The following positions are necessary for contour approach and departure:

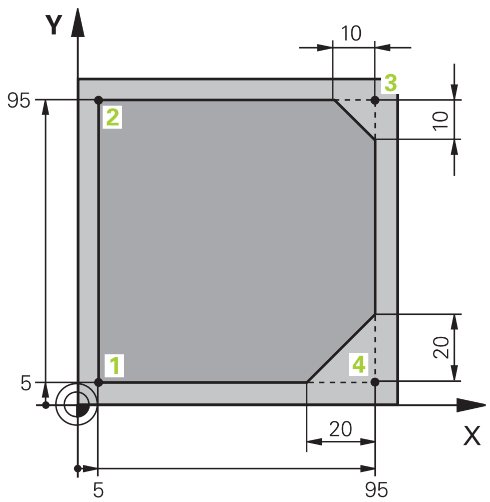
<table>
<tr><th>Help graphic</th><th>Position</th></tr>
<tr><td>
</td><td>Starting point
The following preconditions apply for the starting point:
No tool radius compensation
Approachable without danger of collision
Near to the first contour point
The graphic shows the following information:
If you define the starting point to be in the dark gray area, the contour will be damaged when the first contour point is approached.</td></tr>
<tr><td>
</td><td>Approaching the starting point in the tool axis
Before approaching the first contour point, you must position the tool to the working depth in the tool axis. If there is a danger of collision, approach the starting point in the tool axis separately.</td></tr>
<tr><td></td><td>First contour point
The control moves the tool from the starting point to the first contour point.
You need to program tool radius compensation for the tool movement to the first contour point.</td></tr>
<tr><td>
</td><td>End point
The following preconditions apply for the end point:
Approachable without danger of collision
Near to the last contour point
In order to make sure that the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element
The graphic shows the following information:
If you define the end point to be in the dark gray area, the contour will be damaged when the end point is approached.</td></tr>
</table>

Help graphic	Position
	**Departing from the end point in the tool axis** Program the tool axis separately when departing from the end point.
	Identical starting and end points Do not program any tool radius compensation if the starting point and end point are the same. In order to make sure that the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

More detailed information

- Functions for approaching and departing from the contour
  Further information: "Fundamentals of approach and departure functions", Page 355

4.3.8 Programming a simple contour



Workpiece to be programmed

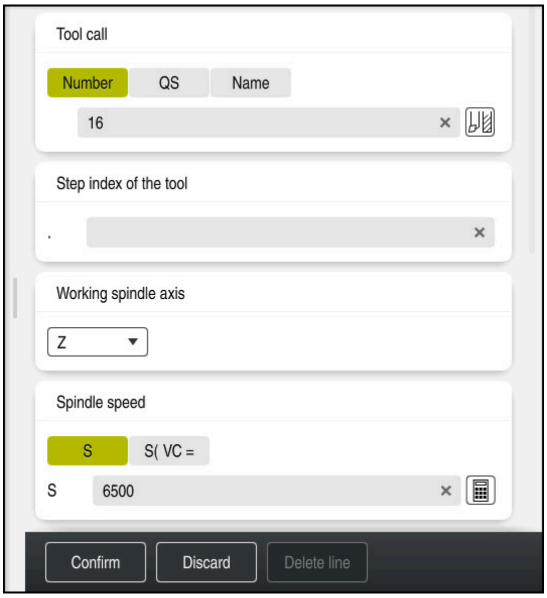
The following texts show you how to mill once at a depth of 5 mm around the contour shown here. You have already defined the workpiece blank.

Further information: "Defining the workpiece blank", Page 133

After you have inserted an NC function, the control shows an explanation about the current syntax element in the dialog bar. You can enter the data directly in the form.

> Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.

Calling a tool



Form column with the syntax elements of the tool call

To call a tool:

TOOL CALL

- Select **TOOL CALL**
- Select **Number** in the form
- Enter the tool number (e.g., **16**)
- Select the tool axis **Z**
- Select the spindle speed **S**
- Enter the spindle speed (e.g., **6500**)

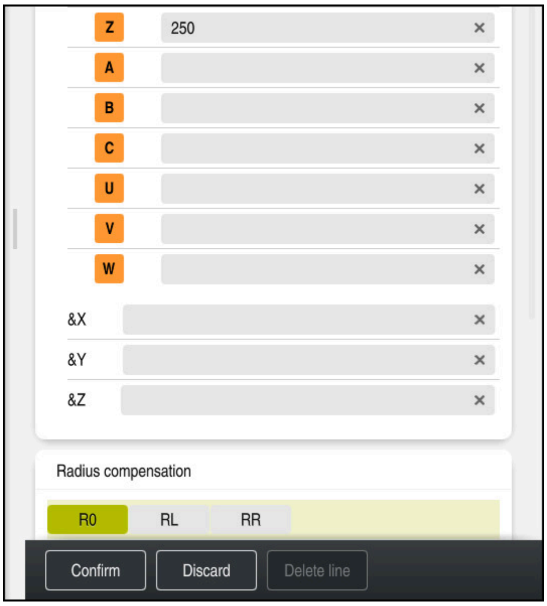
Confirm

- Select **Confirm**
- The control concludes the NC block.

3 TOOL CALL 12 Z S6500

> 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.

Moving the tool to a safe position



Form column with the syntax elements of a straight line

To move the tool to a safe position:

- ▶ Select the path function **L**
- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- > The control applies **R0**, which means there is no tool radius compensation.
- ▶ Select the **FMAX** feed rate
- > The control adopts **FMAX** for rapid traverse.
- ▶ If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)
- ▶ Select **Confirm**
- > The control concludes the NC block.

```
4  L  Z+250 R0 FMAX M3
```

Pre-positioning in the working plane

To pre-position in the working plane:

- ▶ Select the path function **L**
- ▶ Select **X**
- ▶ Enter a value (e.g., **-20**)
- ▶ Select **Y**
- ▶ Enter a value (e.g., **-20**)
- ▶ Select the **FMAX** feed rate
- ▶ Select **Confirm**
- > The control concludes the NC block.

```
5  L  X-20 Y-20 FMAX
```

Pre-positioning in the tool axis

To pre-position in the tool axis:

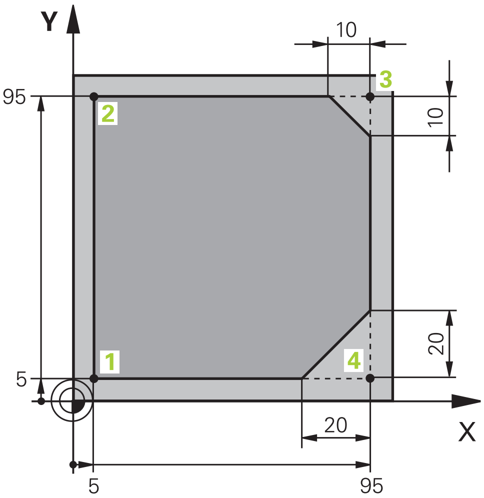
- Select the path function **L**

- Select **Z**
- Enter a value (e.g., **-5**)
- Select the feed rate **F**
- Enter the value for the positioning feed rate (e.g., **3000**)
- If needed, enter a miscellaneous function **M**, such as **M8** (turn coolant on)

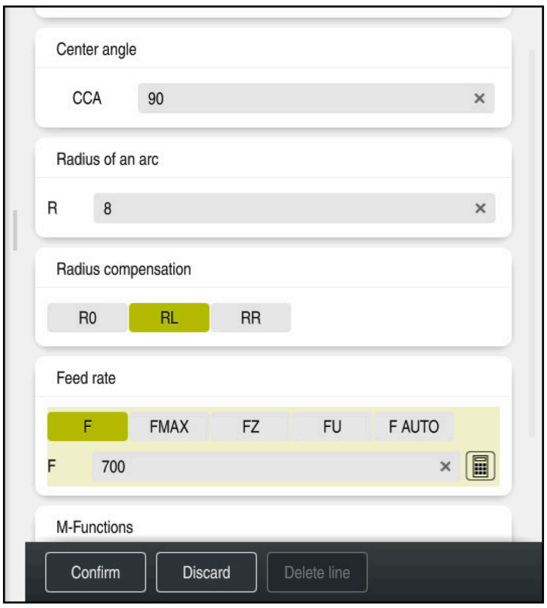
- Select **Confirm**
- > The control concludes the NC block.

```
6 L Z-5 R0 F3000 M8
```

Approaching the contour



Workpiece to be programmed



Form column with the syntax elements of an approach function

To approach the contour:

APPR /DEP

- ▶ Select the **APPR DEP** path function
- > The control opens the **Insert NC function** window.

- ▶ Select **APPR**

- ▶ Select an approach function (e.g., **APPR CT**)

Paste

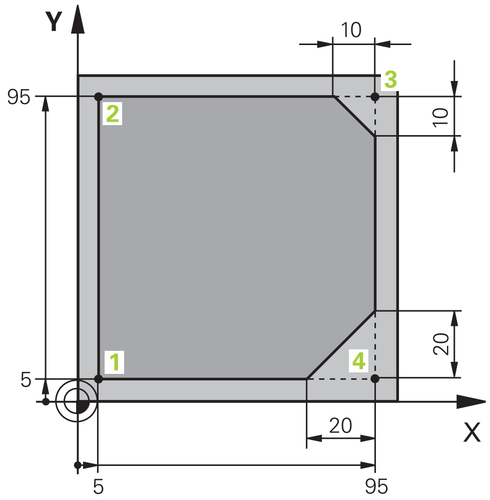
- ▶ Select **Paste**
- ▶ Enter the coordinates of starting point **1** (e.g., **X 5 Y 5**)
- ▶ For the center angle **CCA**, enter the approach angle (e.g., **90**)
- ▶ Enter the radius of the circular arc (e.g., **8**)
- ▶ Select **RL**
- > The control applies tool radius compensation to the left.
- ▶ Select the feed rate **F**
- ▶ Enter the value for the machining feed rate (e.g., **700**)

Confirm

- ▶ Select **Confirm**
- > The control concludes the NC block.

```
7  APPR CT  X+5 Y+5 CCA90 R+8 RL F700
```

Machining a contour



Workpiece to be programmed

To machine the contour:

L
- Select the path function **L**
- Enter the coordinates of contour point 2 that differ (e.g., **Y 95**)

Confirm
- Conclude the NC block with **Confirm**
- > The control applies the changed value and retains all of the other information from the previous NC block.

L
- Select the path function **L**
- Enter the coordinates of contour point 3 that differ (e.g., **X 95**)

Confirm
- Conclude the NC block with **Confirm**

CHF
- Select the path function **CHF**
- Enter the chamfer width (e.g., **10**)

Confirm
- Conclude the NC block with **Confirm**

L
- Select the path function **L**
- Enter the coordinates of contour point 4 that differ (e.g., **Y 5**)

Confirm
- Conclude the NC block with **Confirm**

CHF
- Select the path function **CHF**
- Enter the chamfer width (e.g., **20**)

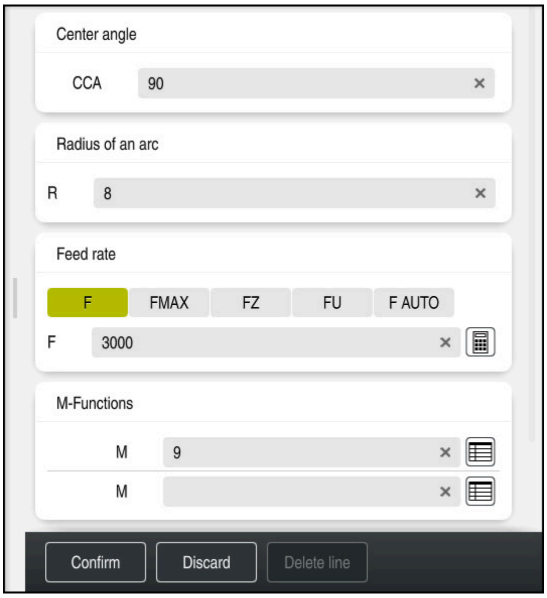
Confirm
- Conclude the NC block with **Confirm**

L
- Select the path function **L**
- Enter the coordinates of contour point 1 that differ (e.g., **X 5**)

Confirm
- Conclude the NC block with **Confirm**

```
8  L  Y+95
9  L  X+95
10  CHF 10
11  L  Y+5
12  CHF 20
13  L  X+5
```

Departing from the contour



Form column with the syntax elements of a departure function

To depart from the contour:

APPR /DEP

- Select the **APPR DEP** path function
- > The control opens the **Insert NC function** window.
- Select **DEP**
- Select a departure function (e.g., **DEP CT**)

Paste

- Select **Paste**
- For the center angle **CCA**, enter the departure angle (e.g., **90**)
- Enter the departure radius (e.g., **8**)
- Select the feed rate **F**
- Enter the value for the positioning feed rate (e.g., **3000**)
- If needed, enter a miscellaneous function **M**, such as **M9** (turn coolant off)

Confirm

- Select **Confirm**
- > The control concludes the NC block.

```
14 DEP CT CCA90 R+8 F3000 M9
```

Moving the tool to a safe position

To move the tool to a safe position:

- ▶ Select the path function **L**
- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- ▶ Select the **FMAX** feed rate
- ▶ Enter a miscellaneous function **M** if required
- ▶ Select **Confirm**
- > The control concludes the NC block.

```
15  L  Z+250 R0 FMAX M30
```

More detailed information

- Tool call
  Further information: "Tool call by TOOL CALL", Page 304
- Line **L**
  Further information: "Straight line L", Page 326
- Designation of the axes and the working plane
  Further information: "Designation of the axes on milling machines", Page 208
- Functions for approaching and departing from the contour
  Further information: "Fundamentals of approach and departure functions", Page 355
- Chamfer **CHF**
  Further information: "Chamfer CHF", Page 328
- Miscellaneous functions
  Further information: "Overview of miscellaneous functions", Page 1319

4.3.9 Programming a machining cycle

The following texts show you how to mill the circular slot of the example task at a depth of 5 mm. You have already defined the workpiece blank and created the outside contour.

Further information: "Example task 1338459", Page 130

After you have inserted a cycle, you can define the associated values in the cycle parameters. You can program the cycle directly in the **Form** column.

Calling a tool

To call a tool:

TOOL CALL

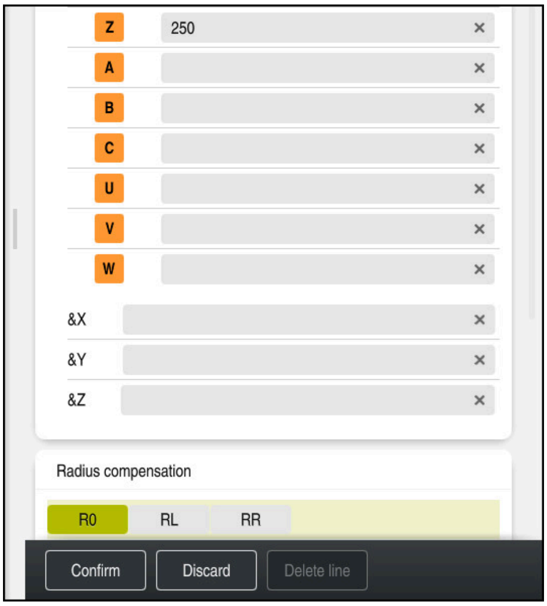
- Select **TOOL CALL**
- Select **Number** in the form
- Enter the tool number (e.g., **6**)
- Select the tool axis **Z**
- Select the spindle speed **S**
- Enter the spindle speed (e.g., **6500**)

Confirm

- Select **Confirm**
- > The control concludes the NC block.

```
16 TOOL CALL 6 Z S6500
```

Moving the tool to a safe position



Form column with the syntax elements of a straight line

To move the tool to a safe position:

L

- Select the path function **L**

Z

- Select **Z**
- Enter a value (e.g., **250**)
- Select tool radius compensation **R0**
- > The control applies **R0**, which means there is no tool radius compensation.
- Select the **FMAX** feed rate
- > The control adopts **FMAX** for rapid traverse.
- If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)

Confirm

- Select **Confirm**
- > The control concludes the NC block.

```
17 L  Z+250 R0 FMAX M3
```

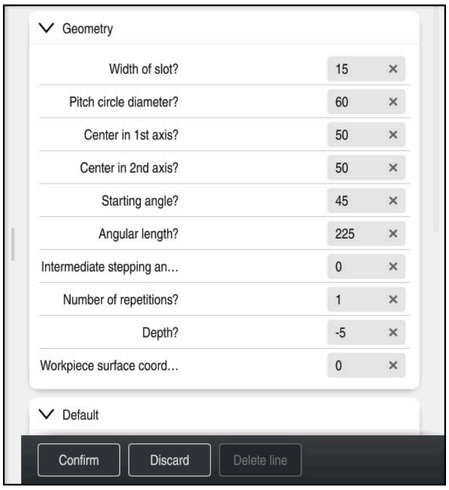
Pre-positioning in the working plane

To pre-position in the working plane:

- Select the path function **L**
- Select **X**
- Enter a value (e.g., **+50**)
- Select **Y**
- Enter a value (e.g., **+50**)
- Select the **FMAX** feed rate
- Select **Confirm**
- > The control concludes the NC block.

```
18 L  X+50 Y+50 FMAX
```

Defining a cycle



Form column with possibilities for entering cycle information

To define the circular slot:

CYCL DEF

- ▶ Select the **CYCL DEF** key
- > The control opens the **Insert NC function** window.

CYCL DEF

- ▶ Select Cycle **254 CIRCULAR SLOT**

Paste

- ▶ Select **Paste**
- > The control inserts the cycle.
- ▶ Open the **Form** column
- ▶ Enter all input values in the form

Confirm

- ▶ Select **Confirm**
- > The control saves the cycle.

```
19 CYCL DEF 254 CIRCULAR SLOT ~
    Q215=+0             ;MACHINING OPERATION ~
    Q219=+15            ;SLOT WIDTH ~
    Q368=+0.1           ;ALLOWANCE FOR SIDE ~
    Q375=+60            ;PITCH CIRCLE DIAMETR ~
    Q367=+0             ;REF. SLOT POSITION ~
    Q216=+50            ;CENTER IN 1ST AXIS ~
    Q217=+50            ;CENTER IN 2ND AXIS ~
    Q376=+45            ;STARTING ANGLE ~
    Q248=+225           ;ANGULAR LENGTH ~
    Q378=+0             ;STEPPING ANGLE ~
    Q377=+1             ;NR OF REPETITIONS ~
    Q207=+500           ;FEED RATE MILLING ~
    Q351=+1             ;CLIMB OR UP-CUT ~
    Q201=-5             ;DEPTH ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q369=+0.1           ;ALLOWANCE FOR FLOOR ~
    Q206=+150           ;FEED RATE FOR PLNGNG ~
    Q338=+5             ;INFEED FOR FINISHING ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q366=+2             ;PLUNGE ~
    Q385=+500           ;FINISHING FEED RATE ~
    Q439=+0             ;FEED RATE REFERENCE
```

Calling a cycle

To call the cycle:

CYCL CALL

- Select **CYCL CALL**

```
20 CYCL CALL
```

Moving the tool to a safe position and concluding the NC program

To move the tool to a safe position:

L

- Select the path function **L**

Z

- Select **Z**
- Enter a value (e.g., **250**)
- Select tool radius compensation **R0**
- Select the **FMAX** feed rate
- Enter a miscellaneous function **M**, such as **M30** (program end)

Confirm

- Select **Confirm**
- > The control concludes the NC block and the NC program.

```
21  L  Z+250 R0 FMAX M30
```

More detailed information

- Machining cycles
  Further information: "Machining Cycles", Page 475
- Calling a cycle
  Further information: "Calling cycles", Page 481

4.3.10 Configuring the control's user interface for simulation

In the **Editor** operating mode you can test NC programs graphically. The control simulates the active NC program in the **Program** workspace.

In order to simulate the NC program you must open the **Simulation** workspace.

> For the simulation you can close the **Form** column to get a better view of the NC program and the **Simulation** workspace.

Opening the Simulation workspace

You can open additional workspaces in the **Editor** operating mode only if an NC program is open.

To open the **Simulation** workspace:

- In the application bar, select **Workspaces**
- Select **Simulation**
- > The control then additionally displays the **Simulation** workspace.

> You can also open the **Simulation** workspace with the **Test Run** operating mode key.

Configuring the Simulation workspace

You can simulate the NC program without needing to enter any special settings. However, an adjustment to the simulation speed is recommended for best viewing of the simulation.

To adjust the speed of the simulation:

- Use the slider to select the factor (e.g., **5.0 * T**)
- > The control then performs the subsequent simulation at five times the speed of the programmed feed rate.

If you use different tables, such as tool tables, for program run and the simulation, then you can define the tables in the **Simulation** workspace.

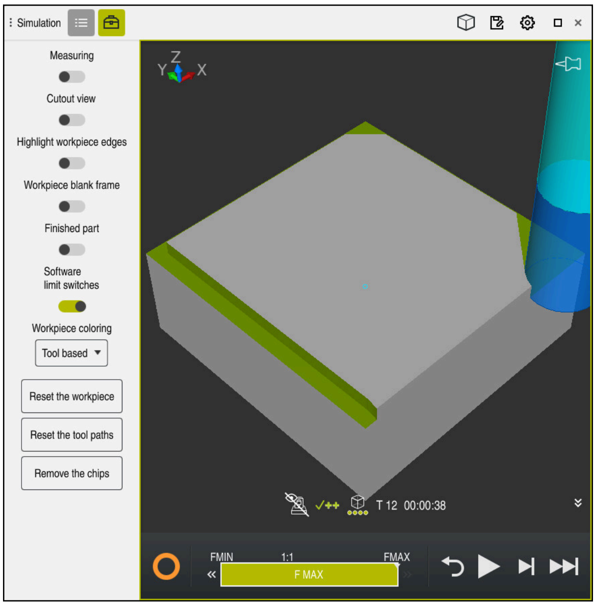
More detailed information

- Workspace: **Simulation**
  Further information: "Simulation Workspace", Page 1535

4.3.11 Simulating an NC program

You test the NC program in the **Simulation** workspace.

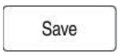
Starting the simulation



Simulation workspace in the **Editor** operating mode

To start the simulation:

- Select **Start**
- > The control asks whether the file should be saved.



- Select **Save**
- > The control starts the simulation.
- > The control uses the **Control-in-operation** symbol to show the simulation status.

Definition

Control-in-operation:
The control uses the **Control-in-operation** symbol to show the current simulation status in the action bar and on the tab of the NC program:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

More detailed information

- **Simulation** workspace
  Further information: "Simulation Workspace", Page 1535

4.4 Configuring a tool

4.4.1 Selecting the Tables operating mode

You configure tools in the **Tables** operating mode.

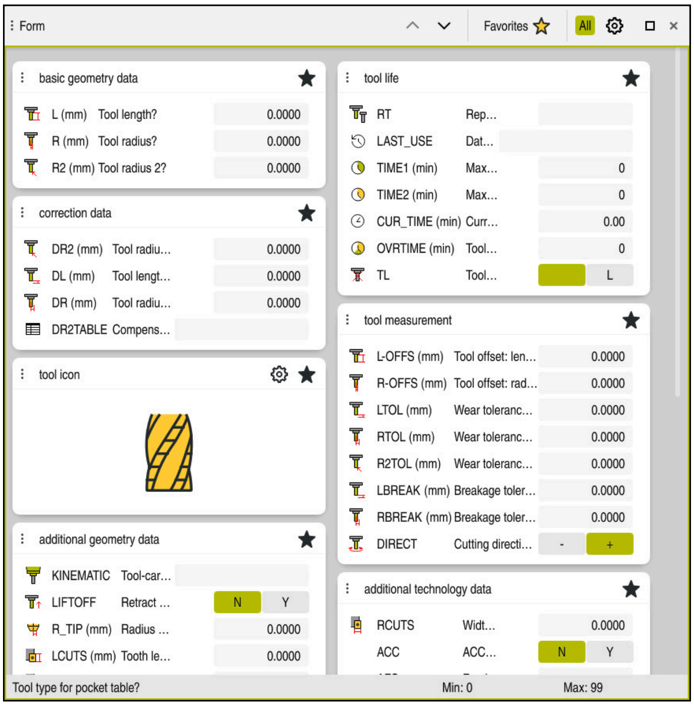
To select the **Tables** operating mode:

- ▶ Select the **Tables** operating mode
- > The control displays the **Tables** operating mode.

More detailed information

- Operating mode: **Tables**
  Further information: "Tables operating mode", Page 1980

4.4.2 Configuring the control's user interface



Form workspace in the **Tables** operating mode

In the **Tables** operating mode you open and edit the various tables of the control either in the **Table** workspace or in the **Form** workspace.

> **i** The first steps describe the procedure with the **Form** workspace open.

To open the **Form** workspace:

- ▶ In the application bar, select **Workspaces**
- ▶ Select **Form**
- > The control opens the **Form** workspace.

More detailed information

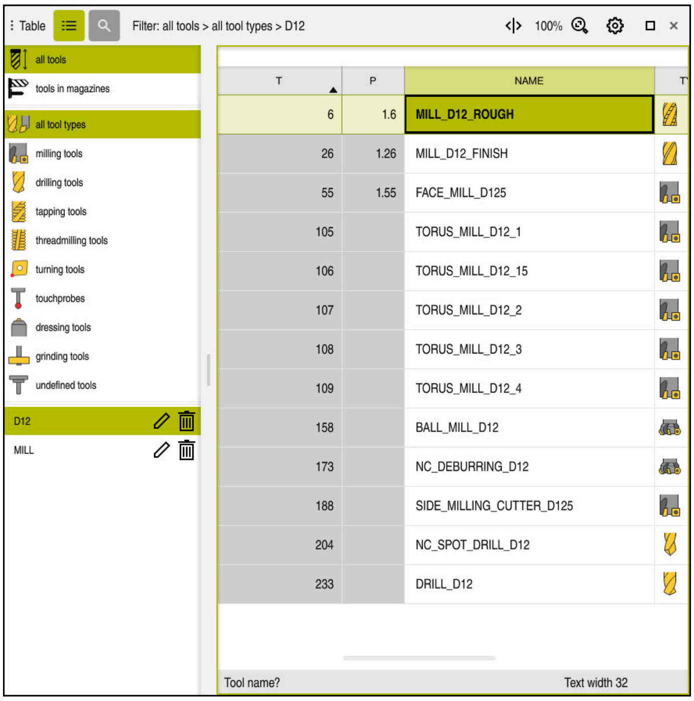
- **Form** workspace
  Further information: "Form workspace for tables", Page 1989
- Workspace: **Table**
  Further information: "Table workspace", Page 1982

4.4.3 Preparing and measuring tools

To prepare tools:

- Clamp the required tools in their tool holders
- Measure the tools
- Write down the length and the radius or transfer these directly to the control

4.4.4 Editing within tool management



Tool management application in the **Table** workspace

Tool management allows you to save tool data, such as the length and radius as well as other tool-specific information.

The control displays the tool data for all tool types in tool management. In the **Form** workspace the control displays only the relevant tool data for the current tool type.

To enter the tool data in tool management:

- Select **Tool management**
- > The control displays the **Tool management** application.
- Open the **Form** workspace

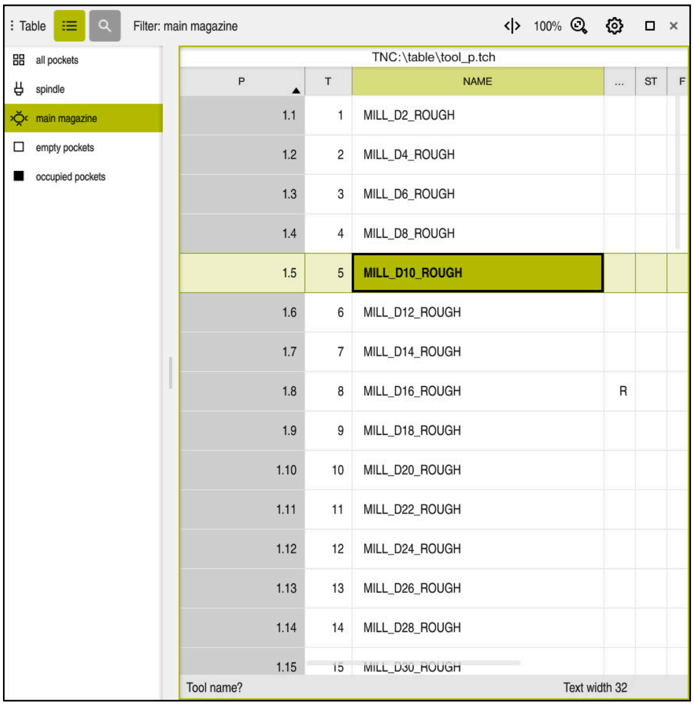
- Activate **Edit**
- Select the desired tool number (e.g., **16**)
- > The control displays the tool data of the selected tool in the form.
- Define the required tool data in the form; for example, the length **L** and the tool radius **R**

More detailed information

- Operating mode: **Tables**
  Further information: "Tables operating mode", Page 1980
- Workspace: **Form**
  Further information: "Form workspace for tables", Page 1989
- Tool management
  Further information: "Tool management ", Page 297
- Tool types
  Further information: "Tool types", Page 281

4.4.5 Editing the pocket table

> Refer to your machine manual!
> Access to the **tool_p.tch** pocket table is machine-dependent.



Pocket table application in the **Table** workspace

The control assigns a pocket in the tool magazine to each tool that is in the tool table. This assignment, as well as the load situation of each tool, is shown in the pocket table.

There are various ways of accessing the pocket table:

- Functions of the machine manufacturer
- Third-party tool-management system
- Manual access to the control

To enter the data in the pocket table:

- ▶ Select **Pocket table**
- > The control displays the **Pocket table** application.
- ▶ Open the **Form** workspace

- ▶ Activate **Edit**
- ▶ Select the desired pocket number
- ▶ Define the tool number
- ▶ Define any additional tool data if necessary, such as whether the pocket is reserved

More detailed information

- Pocket table

  Further information: "Pocket table tool_p.tch", Page 2026

4.5 Setting up a workpiece

4.5.1 Selecting an operating mode

You set up workpieces in the **Manual** operating mode.

To select the **Manual** operating mode:

- Select the **Manual** operating mode
- > The control displays the **Manual** operating mode.

More detailed information

- Operating mode: **Manual**
  Further information: "Overview of operating modes", Page 110

4.5.2 Clamping the workpiece

Mount the workpiece with a fixture on the machine table.

4.5.3 Workpiece presetting with a touch probe

Inserting a workpiece touch probe

Use a workpiece touch probe to set up the workpiece with the aid of the control and set the workpiece preset.

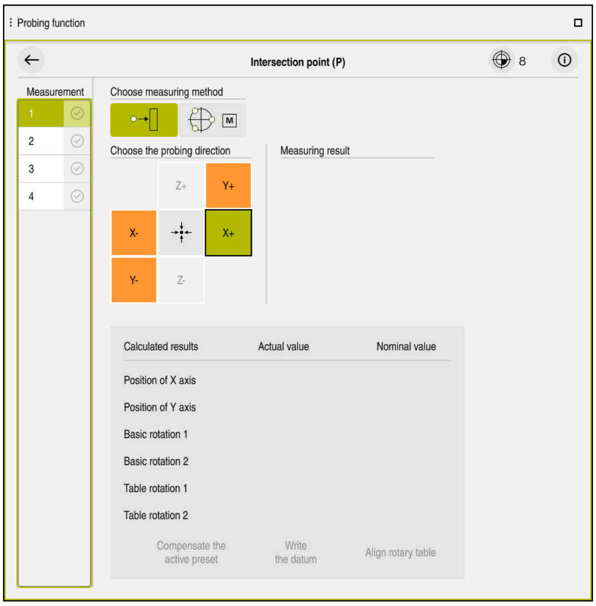
To insert a workpiece touch probe:

- Select **T**
- Enter the tool number of the workpiece touch probe, (e.g., **600**)
- Press the **NC Start** key
- > The controls inserts the workpiece touch probe.

Setting a workpiece preset

To set a workpiece preset at a corner:

- Select the **Setup** application
  - Select **Intersection point (P)**
  - > The control opens the probing cycle.
  - Manually position the touch probe near the first touch point of the first workpiece edge
  - In the **Choose the probing direction** area, select the direction of probing (e.g., **Y+**)
  - Press the **NC Start** key
  - > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
  - Manually position the touch probe near the second touch point of the first workpiece edge
  - Press the **NC Start** key
  - > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
  - Manually position the touch probe near the first touch point of the second workpiece edge
  - In the **Choose the probing direction** area, select the direction of probing (e.g., **X+**)
  - Press the **NC Start** key
  - > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
  - Manually position the touch probe near the second touch point of the second workpiece edge
  - Press the **NC Start** key
  - > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
  - > The control then displays the coordinates of the determined corner point in the **Measuring result** area.
  - Select **Compensate the active preset**
  - > The control applies the calculated results to the workpiece preset.
  - > The control highlights the line with a preset symbol.
  - Select **Exit probing**
  - > The control closes the probing cycle.



Probing function workspace with an open manual probing function

More detailed information

- Workspace: **Probing function**

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557
- Reference points in the machine

  Further information: "Presets in the machine", Page 210
- Tool change in the **Manual operation** application

  Further information: "Manual operation application", Page 202

4.6 Machining a workpiece

4.6.1 Selecting an operating mode

You machine workpieces in the **Program Run** operating mode.

To select the **Program Run** operating mode:

- Select the **Program Run** operating mode
- > The control displays the **Program Run** operating mode and the most recently executed NC program.

More detailed information

- Operating mode: **Program Run**
  Further information: "Program Run operating mode", Page 1954

4.6.2 Opening an NC program

To open an NC program:

- Select **Open File**
- > The control displays the **Open File** workspace.
- Select an NC program
- Select **Open**
- > The control opens the NC program.

More detailed information

- Workspace: **Open File**
  Further information: "Open File workspace", Page 1153

4.6.3 Starting an NC program

To start an NC program:

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

4.7 Switching the machine off

Refer to your machine manual.
Switching off is a machine-dependent function.

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss regardless of the control's status!

- Always shut down the control
- Only operate the main switch after being prompted on the screen

To power-off the machine:

- Select the **Home** operating mode

Shut down

- Select **Shut down**
- > The control opens the **Shut down** window.

Shut down

- Select **Shut down**
- > If NC programs or contours contain any unsaved changes, the control displays the **Close the program** window.
- If necessary, save unsaved NC programs with **Save** or **Save as**
- > The control shuts down.
- > Once shutdown has concluded, the control displays the text **Now you can switch off.**
- Switch off the main power switch of the machine

5

Status Displays

5.1 Overview

The control shows the status or values of individual functions in the status displays.

The control offer the following status displays:

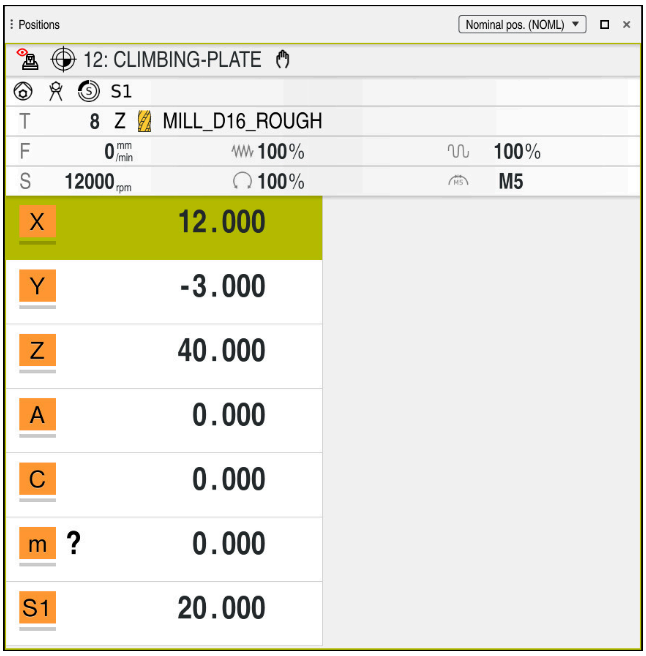
- General status display and position display in the **Positions** workspace
  Further information: "Positions workspace", Page 163
- Status overview on the TNC bar
  Further information: "Status overview on the TNC bar", Page 169
- Additional status displays for specific areas in the **Status** workspace
  Further information: "Status workspace", Page 171
- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece
  Further information: "Simulation status workspace", Page 186

5.2 Positions workspace

Application

The general status display in the **Positions** workspace provides information about the status of various functions of the control and about current axis positions.

Description of function



Positions workspace with general status display

You can open the **Positions** workspace in the following operating modes:

- **Manual**
- **Program Run**

Further information: "Overview of operating modes", Page 110

The **Positions** workspace provides the following information:

- Symbols of active and inactive functions, such as Dynamic Collision Monitoring (DCM, option 40)
- Active tool
- Technology values
- Settings of the spindle and feed-rate potentiometers
- Active miscellaneous functions for the spindle
- Axis values and statuses, such as "Axis not referenced"

  Further information: "Test status of the axes", Page 2096

Axis display and position display

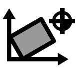
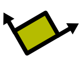
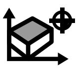
Refer to your machine manual.
In the machine parameter **axisDisplay** (no. 100810) you define the quantity and sequence of the displayed axes.

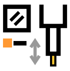
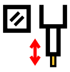
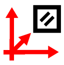
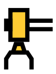
Icon	Meaning
ACTL	Position display mode (e.g., actual or nominal coordinates of the current tool position) You can select the mode in the title bar of the workspace. **Further information:** "Position displays", Page 188
X m	Axes The X axis is selected. You can move the selected axis. The auxiliary axis **m** is not selected. The control displays auxiliary axes, such as the tool magazine, as lowercase letters. **Further information:** "Definition", Page 168
?	The axis is not referenced.
	The axis is not in safe mode. **Further information:** "Checking axis positions manually", Page 2097
Δ	The axis is moving the distance-to-go shown next to the symbol.
	The axis is clamped.
	You can move the axis with the handwheel.
F_{SOS} 0 mm/min	Feed rate when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 2093
S_{SOS} 0 U/min	Spindle when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 2093

Presets and technology values

Symbol	Meaning
	Number and comment of the active workpiece preset The number corresponds to the active row number of the preset table. The comment corresponds to the content of the **DOC** column. **Further information:** "Preset management", Page 1025
T	In the **T** area, the control shows the following information: ■ Number of the active tool ■ Tool axis of the active tool ■ Symbol of the defined tool type ■ Name of the active tool
F	In the **F** area, the control shows the following information: ■ Active feed rate in mm/min You can program the feed rate in various units of measurement. The control always converts the programmed feed rate in this display to mm/min. ■ Setting of the rapid-traverse potentiometer in percent ■ Setting of the feed-rate potentiometer in percent **Further information:** "Potentiometers", Page 122 If a feed rate limit is active using the **F MAX** button, the area is called **FMAX** instead of **F**. The control displays the text **FMAX** and the feed rate value in orange. **Further information:** "Feed rate limit FMAX", Page 1958
S	In the **S** area, the control shows the following information: ■ Active shaft speed in rpm If you have programmed a cutting speed instead of a rotational speed, the control automatically converts this value to a rotational speed. ■ Setting of the spindle potentiometer in percent ■ Active miscellaneous function for the spindle

Active functions

Icon	Meaning
	The **Manual traverse** function is active.
	The **Manual traverse** function is not active. **Further information:** "Program Run operating mode", Page 1954
	**RL** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1114
	**RR** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1114 These icons are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1965
	**R+** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1114
	**R-** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1114 These icons are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1965
	3D tool compensation is active. **Further information:** "3D tool compensation (option 9)", Page 1126 This icon is transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1965
	A basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 1027
	The basic rotation will be taken into account while moving the axes. **Further information:** "Basic rotation selection item", Page 1100
	A 3D basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 1027

Icon	Meaning
	The tilted working plane will be taken into account while moving the axes. **Further information:** "Tilting the working plane with PLANE functions (option 8)", Page 1054 **Further information:** "3D ROT selection item", Page 1101
	The **Tool axis** function is active. **Further information:** "Tool axis selection item", Page 1101
	Either the **TRANS MIRROR** function or Cycle **8 MIRRORING** is active. The axes programmed in the function or cycle are mirrored and moved. **Further information:** "Cycle 8 MIRRORING", Page 1036 **Further information:** "Mirroring with TRANS MIRROR", Page 1047
	The pulsing spindle speed function **S-PULSE** is active. **Further information:** "Pulsing spindle speed with FUNCTION S-PULSE", Page 1206
	The **PARAXCOMP DISPLAY** function is active.
	The **PARAXCOMP MOVE** function is active. **Further information:** "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1284
	The **PARAXMODE** function is active. This icon might be superimposed on the icons for **PARAX-COMP DISPLAY** and **PARAXCOMP MOVE**. **Further information:** "Select three linear axes for machining with FUNCTION PARAXMODE", Page 1289
TCPM	Either the **M128** function or **TCPM FUNCTION** (option 9) is active. **Further information:** "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
	The turning mode **FUNCTION MODE TURN** (option 50) is active. **Further information:** "Switching the operating mode with FUNCTION MODE", Page 234
	The grinding mode **FUNCTION MODE GRIND** (option 156) is active. **Further information:** "Switching the operating mode with FUNCTION MODE", Page 234
	Dressing mode (option 156) is active. **Further information:** "Activating dressing mode with FUNCTION DRESS", Page 254
	Dynamic Collision Monitoring (DCM, option 40) is active.

Icon	Meaning
	Dynamic Collision Monitoring (DCM, option 40) is not active. **Further information:** "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
AFC L	Adaptive Feed Control (AFC, option 45) is active in teach-in cut mode.
AFC	Adaptive Feed Control (AFC, option 45) is active in closed-loop mode. **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 1196
ACC	Active Chatter Control (ACC, option 145) is active. **Further information:** "Active Chatter Control (ACC, option 145)", Page 1204
	Global Program Settings (GPS, option 44) are active. **Further information:** "Global Program Settings (GPS, option 44)", Page 1217
	Process Monitoring (option 168) is active. **Further information:** "Process Monitoring (option 168)", Page 1236

In the optional machine parameter **iconPrioList** (no. 100813) you can change the sequence in which the control displays these symbols. The symbol for Dynamic Collision Monitoring (DCM, option 40) is always visible and cannot be configured.

Definition

Auxiliary axes
Auxiliary axes are controlled by the PLC and are not included in the kinematics description. Auxiliary axes are driven, for example, hydraulically, electrically, or by an external motor. The machine manufacturer can define the tool magazine, for example, as an auxiliary axis.

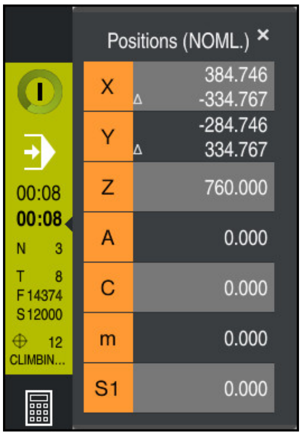
5.3 Status overview on the TNC bar

Application

On the TNC bar, the control shows a status overview with the execution status, the current technology values, and the axis positions.

Description of function

General information



When you are machining an NC program or individual NC blocks, the control provides the following information on the TNC bar:

- **Control-in-operation**: current machining status
  Further information: "Definition", Page 170
- Symbol of the application used for machining
- Remaining run time of the NC program
- Program run time
  The control displays the run times of the NC program in mm:ss format. As soon as an NC program run time exceeds 59:59, the control changes the format to hh:mm.

> The control displays the same value for the program run time as on the **PGM** tab of the **Status** workspace.
> In the **Status** workspace the control shows the program run time in hh:mm:ss format.
> **Further information:** "Display of the program run time", Page 187

- Active tool
- Active feed rate
- Current spindle speed
- Number and comment of the active workpiece preset

Position display

If you select the status overview area, the control opens or closes the position display with the current axis positions. The control uses the same position display mode as in the **Positions** workspace, for example **Actual pos. (ACT)**.

Further information: "Positions workspace", Page 163

If you select an axis line, the control copies the current value of this line to the clipboard.

Press the **actual position capture** key to open the position display. The control prompts you to select the value to be copied to the clipboard. During programming, you can thus transfer the values directly into a programming dialog.

Definition

Control-in-operation:
The control uses the **Control-in-operation** symbol to show the machining status of the NC program or NC block:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

Further information: "Interrupting, stopping or canceling program run", Page 1959

When the control bar is expanded, the control shows additional information about the current status, such as **Active, feed rate at zero**.

5.4 Status workspace

Application

In the **Status** workspace the control shows the additional status display. The additional status display shows the current status of various functions on specific tabs. You can use the additional status display to better monitor the running of an NC program by receiving real-time information about active functions and accesses.

Description of function

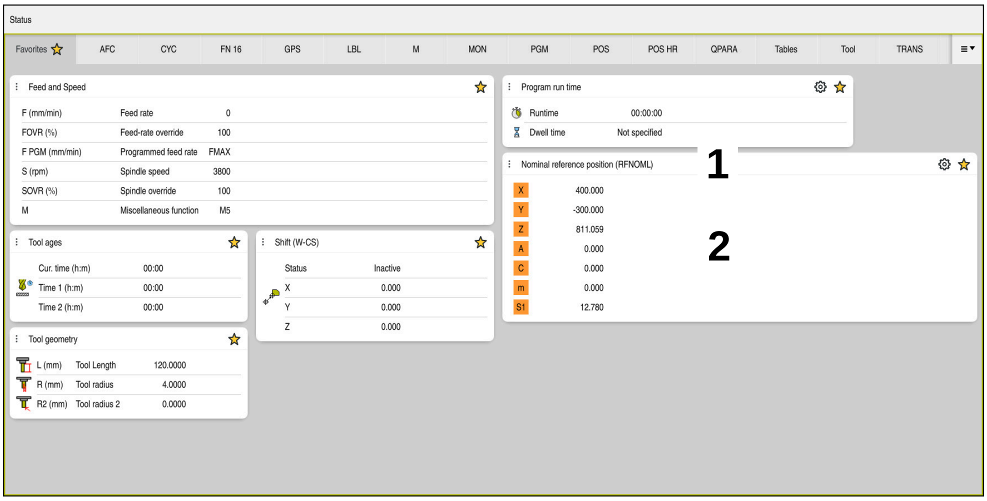
You can open the **Status** workspace in the following operating modes:

- **Manual**
- **Program Run**

Further information: "Overview of operating modes", Page 110

Favorites tab

On the **Favorites** tab you can arrange your own status display with contents from the other tabs.



Favorites tab

1 Area
2 Contents

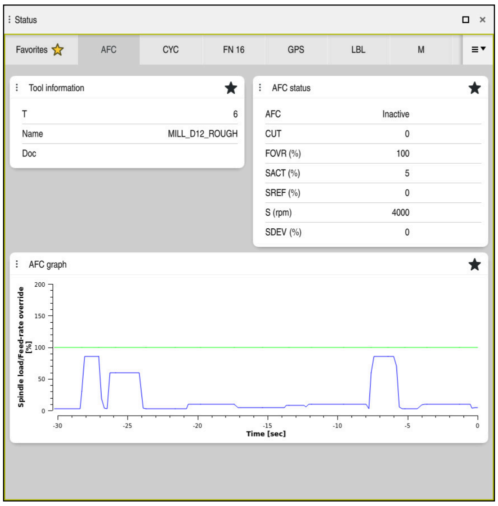
Every area in the status display has a **Favorites** icon. If you select the icon, the control adds that area to the **Favorites** tab.

Further information: "Icons on the control's user interface", Page 123

AFC tab (option 45)

The control displays information about the Adaptive Feed Control function (AFC, option 45) on the **AFC** tab.

Further information: "Adaptive Feed Control (AFC, option 45)", Page 1196



AFC tab

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Comment about the tool from the tool management
AFC status	■ **AFC** If AFC is being used to control the feed rate, then **Control** is displayed in this area. If the control is not controlling the feed rate, then **Inactive** is displayed in this area. ■ **CUT** Counts the quantity of cuts that have been performed with **FUNCTION AFC CUT BEGIN**, starting from zero. ■ **FOVR (%)** Active factor of the feed-rate potentiometer in percent ■ **SACT (%)** Current spindle load in percent ■ **SREF (%)** Reference load of the spindle in percent Define the reference load of the spindle in the syntax element **LOAD** of the **FUNCTION AFC CUT BEGIN** function. **Further information:** "NC functions for AFC (option 45)", Page 1199 ■ **S (rpm)** Spindle shaft speed in rpm ■ **SDEV (%)** Current deviation of the speed in percent

Area	Contents
AFC graph	The **AFC graph** visualizes the relationship between the elapsed **Time [sec]** and the **Spindle load/Feed-rate override [%]**. The green line in the graph shows the feed-rate override and the blue line shows the spindle load.

CYC tab

On the **CYC** tab the control shows information about machining cycles.

Area	Contents
Active cycle definition	When you use the **CYCLE DEF** function to define a cycle, the control shows the cycle number in this area.
Cycle 32 TOLERANCE	▪ **Status** Shows whether Cycle **32 TOLERANCE** is active or inactive ▪ Values of Cycle **32 TOLERANCE** ▪ Values from the machine manufacturer for path and angle tolerance, such as predefined machine-specific roughing or finishing filters ▪ Values of Cycle **32 TOLERANCE** that are restricted by Dynamic Collision Monitoring (DCM, option 40)

The machine manufacturer uses Dynamic Collision Monitoring (DCM, option 40) to define the restriction of the tolerance.

In the optional machine parameter **maxLinearTolerance** (no. 205305) the machine manufacturer defines the maximum permissible linear tolerance. In the optional machine parameter **maxAngleTolerance** (no. 205303) the machine manufacturer defines the maximum permissible angle tolerance. If DCM is active, the control restricts the tolerance defined in **32 TOLERANCE** to these values.

If the tolerance is restricted by DCM, the control displays a gray warning triangle as well as the restricted values.

FN16 tab

On the **FN16** tab the control displays the contents of a file that was output with **FN 16: F-PRINT**.

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1382

Area	Contents
Output	Contents of an output file that was output with **FN 16: F-PRINT**, such as measured values or texts.

GPS tab (option 44)

The control displays information about the Global Program Settings (GPS, option 44) on the **GPS** tab.

Further information: "Global Program Settings (GPS, option 44)", Page 1217

Area	Contents
Additive offset (M-CS)	▪ **Status** The **Status** shows whether a function is active or inactive. A function can be active even if its values are zero. ▪ **A (°)** **Additive offset (M-CS)** in the A axis The **Additive offset (M-CS)** function is also available for the other rotary axes **B (°)** and **C (°)**.
Additive basic rotat. (W-CS)	▪ **Status** ▪ **(°)** The **Additive basic rotat. (W-CS)** function is active in the workpiece coordinate system **W-CS**. Entries are in degrees. **Further information:** "Workpiece coordinate system W-CS", Page 1016
Shift (W-CS)	▪ **Status** ▪ **X** **Shift (W-CS)** in the X axis The **Shift (W-CS)** function is also available for the other linear axes **Y** and **Z**.
Mirroring (W-CS)	▪ **Status** ▪ **X** **Mirroring (W-CS)** in the X axis The **Mirroring (W-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.
Rotation (WPL-CS)	▪ **Status** ▪ **(°)** **Rotation (WPL-CS)** in degrees The **Rotation (WPL-CS)** function is active in the working plane coordinate system **WPL-CS**. Entries are in degrees. **Further information:** "Working plane coordinate system WPL-CS", Page 1018
Shift (mW-CS)	▪ **Status** ▪ **X** **Shift (mW-CS)** in the X axis The **Shift (mW-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.

Area	Contents
Handwheel superimp.	▪ **Status** ▪ **Coordinate system** This area contains the selected coordinate system for **Handwheel superimp.**, such as the machine coordinate system **M-CS**. ▪ **X** ▪ **Y** ▪ **Z** ▪ **A (°)** ▪ **B (°)** ▪ **C (°)** ▪ **VT**
Feed rate factor	If the **Feed rate factor** function is active, the control displays the defined percentage in this field. If the **Feed rate factor** function is not active, the control displays **100.00 %** in this field.

LBL tab

On the **LBL** tab the control shows information about program section repeats and subprograms.

Further information: "Subprograms and program section repeats with the label LBL", Page 384

Area	Contents
Subprogram calls	▪ **Blk. no.** Block number of the call ▪ **LBL no./Name** Called label
Repetitions	▪ **Blk. no.** ▪ **LBL no./Name** ▪ **Program-section repeat** Number of repetitions still to be performed, e.g. 4/5

M tab

On the **M** tab the control shows information about active miscellaneous functions.

Further information: "Miscellaneous Functions", Page 1317

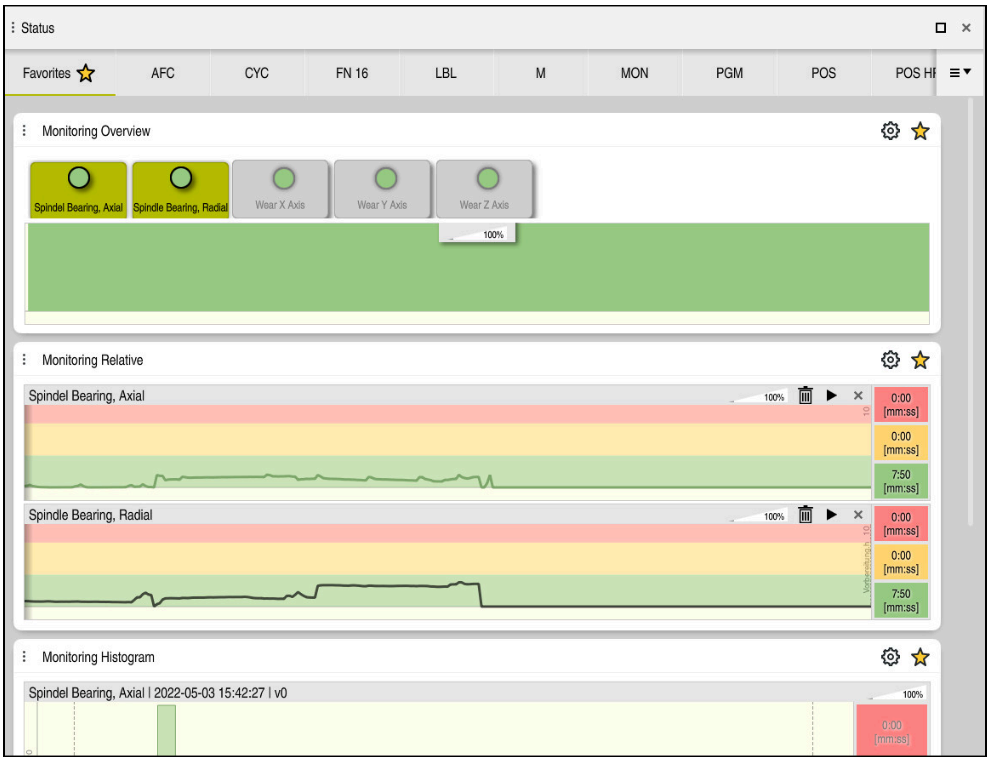
Area	Contents
Active M functions	■ **Function** Active miscellaneous functions, such as **M3** ■ **Description** Descriptive text about the respective miscellaneous function. Refer to your machine manual. Only the machine manufacturer can create a descriptive text for machine-specific miscellaneous functions.

MON tab (option 155)

On the **MON** tab the control shows information about machine components defined to be monitored through Component Monitoring (option 155).

Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1230

> Refer to your machine manual.
> The machine manufacturer specifies which machine components are monitored, and to what extent.



MON tab with configured spindle speed monitoring

Area	Contents
Monitoring Overview	The control displays the machine components defined for monitoring. By selecting a component, you hide or show whether it is being monitored.
Monitoring Relative	The control displays the monitoring information for the components being shown in the **Monitoring Overview** area. ■ Green: component works under conditions defined as safe ■ Yellow: component works under warning zone conditions ■ Red: component is overloaded In the **Display settings** window you can select which component the control shows.
Monitoring Histogram	The control shows a graphical evaluation of previous monitoring sessions.

Use the **Settings** symbol to open the **Display settings** window. You can define the height of each graphical depiction for each area.

PGM tab

On the **PGM** tab the control shows information about the program run.

Area	Contents
Parts counter	■ **Quantity** Actual value and nominal value of the parts counter defined with the **FUNCTION COUNT** function **Further information:** "Defining counters with FUNCTION COUNT", Page 1409
Program run time	■ **Runtime** Run time of the NC program in hh:mm:ss format ■ **Dwell time** Countdown of the waiting time in seconds from the following functions: ■ **FUNCTION DWELL** ■ Cycle **9 DWELL TIME** ■ Parameter **Q210 DWELL TIME AT TOP** ■ Parameter **Q211 DWELL TIME AT DEPTH** ■ Parameter **Q255 DWELL TIME** **Further information:** "Display of the program run time", Page 187
Programs called	Path of the main program as well as called NC programs including the path
Pole/circle center	Programmed axes and values of the circle center point **CC**
Radius compensation	Programmed tool radius compensation

POS tab

On the **POS** tab the control shows information about positions and coordinates.

Area	Contents
Position display, e.g., **Actual reference position (RFACTL)**	In this area the control shows the current position of all axes that are present. You can choose between the following views in the position display: ■ **Nominal pos. (NOML)** ■ **Actual pos. (ACT)** ■ **Nominal reference position (RFNOML)** ■ **Actual reference position (RFACTL)** ■ **Servo lag (LAG)** ■ **Handwheel superimposed (M118)** **Further information:** "Position displays", Page 188
Feed and Speed	■ Active **Feed** in mm/min If a feed rate limit is active, the control displays the line in orange. If the feed rate is limited using the **FMAX** button, the control displays **MAX** in square brackets. **Further information:** "Feed rate limit FMAX", Page 1958 If the feed rate is limited using the **F limited** button, the control displays the active safety function in square brackets. **Further information:** "Safety functions", Page 2092 ■ Active **Feed-rate override** in % ■ Active **Rapid-traverse override** in % ■ Active **Programmed feed rate** in mm/min ■ Active **Spindle speed** in rpm ■ Active **Spindle override** in % ■ Active **Miscellaneous function** in reference to the spindle, such as **M3**
Orientation of the working plane	Spatial angles or axis angles for the active working plane **Further information:** "Tilting the working plane with PLANE functions (option 8)", Page 1054 If axis angles are active, the control displays in this area only the values of the physically present axes. The defined values are in the window **3-D rotation** **Further information:** "3D ROT selection item", Page 1101
OEM transformation	The machine manufacturer can define an OEM transformation for special turning kinematics. **Further information:** "Definitions", Page 185
Basic transformations	In this area the control shows the values of the active workpiece preset and active transformations in linear and rotary axes, such as a transformation in the X axis with the function **TRANS DATUM**. **Further information:** "Preset management", Page 1025

Area	Contents
Special turning transformations	Transformations relevant for turning operations (option 50), such as a defined **precession angle** from the following sources: ■ Defined by the machine manufacturer ■ Cycle **800 ADJUST XZ SYSTEM** ■ Cycle **801 RESET ROTARY COORDINATE SYSTEM** ■ Cycle **880 GEAR HOBBING**
Active traverse ranges	Active traverse range, such as Limit 1 for traverse range 1 Traverse ranges are machine-specific. If no traverse range is active, then **Traverse range not defined** is displayed in this area.
Active kinemat.	Name of the active machine kinematics

POS HR tab

On the **POS HR** tab the control shows information about handwheel superimpositioning.

Area	Contents
Coordinate system	■ **Machine (M-CS)** With **M118** the handwheel superimpositioning is always in effect in the machine coordinate system **M-CS**. **Further information:** "Activating handwheel superimpositioning with M118", Page 1334 With the Global Program Settings (GPS, option 44) the coordinate system can be chosen. **Further information:** "Global Program Settings (GPS, option 44)", Page 1217
Handwheel superimp.	■ **Max. val.** Maximum value of the individual axes programmed in **M118** or in the **GS** workspace ■ **Actl.val.** Current superimpositioning

QPARA tab

On the **QPARA** tab the control shows information about the defined variables.

Further information: "Variables: Q, QL, QR and QS parameters", Page 1362

You use the **Parameter list** window to define which variables the control shows in the areas.

Further information: "Defining the contents of the QPARA tab", Page 191

Area	Contents
Q parameter	Shows the values of the selected Q parameters
QL parameter	Shows the values of the selected QL parameters
QR parameter	Shows the values of the selected QR parameters
QS parameter	Shows the contents of the selected QS parameters

Tables tab

On the **Tables** tab the control shows information about the active tables for program run or the simulation.

Area	Contents
Active tables	In this area the control shows the path for the following active tables: ■ Tool table ■ Turning tool table ■ Preset table ■ Datum table ■ Pocket table ■ Touch probe table ■ Grinding tool table ■ Dressing tool table

TRANS tab

On the **TRANS** tab the control shows information about active transformations in the NC program.

Area	Contents
Active datum	■ Path of the selected datum table ■ Row number of the selected datum table ■ **Doc** Contents of the **DOC** column of the datum table
Active datum shift	Datum shift that was defined with the **TRANS DATUM** function **Further information:** "Datum shift with TRANS DATUM", Page 1046
Mirrored axes	Axes mirrored with either the **TRANS MIRROR** function or Cycle **8 MIRRORING** **Further information:** "Mirroring with TRANS MIRROR", Page 1047 **Further information:** "Cycle 8 MIRRORING", Page 1036
Active angle of rotation	Rotation angle defined with either the **TRANS ROTATION** function or Cycle **10 ROTATION** **Further information:** "Rotations with TRANS ROTATION", Page 1050 **Further information:** "Cycle 10 ROTATION ", Page 1038
Orientation of the working plane	Spatial angles or axis angles for the active working plane **Further information:** "Tilting the working plane with PLANE functions (option 8)", Page 1054
Center coordinates	Center of scaling that was defined with Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** "Cycle 26 AXIS-SPECIFIC SCALING ", Page 1041

Area	Contents
Active scaling factors	Scaling factors that were defined for the individual axes with the **TRANS SCALE** function, Cycle **11 SCALING FACTOR** or Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** "Scaling with TRANS SCALE", Page 1051 **Further information:** "Cycle 11 SCALING FACTOR ", Page 1040 **Further information:** "Cycle 26 AXIS-SPECIFIC SCALING ", Page 1041
Shift (WPL-CS)	Active shift in the working plane coordinate system **WPL-CS** using the following function: ■ **FUNCTION CORRDATA** **Further information:** "Activating a compensation value with FUNCTION CORRDATA", Page 1123 ■ **FUNCTION TURNDATA CORR** (option 50) **Further information:** "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1124
Table	■ Path of the selected compensation table ***.wco** ■ Row number of the selected compensation table ***.wco** ■ Content of the **DOC** column of the active row **Further information:** "Compensation table *.wco", Page 2057

TT tab

On the **TT** tab the control shows information about measurements performed with a TT tool touch probe.

Further information: "Hardware enhancements", Page 107

<table>
<tr><th>Area</th><th>Contents</th></tr>
<tr><td>TT Tool to be measured</td><td>

T
Tool number
Name
Tool name
Measure what?
Measurement method chosen for tool measurement, such as Length
Min (mm)
When measuring milling cutters, in this area the control shows the smallest measured value of a cutting edge.
When measuring turning tools (option 50), in this area the control shows the smallest measured tipping angle. The value of the angle can also be negative.
Further information: "Definitions", Page 185
Max (mm)
When measuring milling cutters, in this area the control shows the greatest measured value of a cutting edge.
When measuring turning tools, in this area the control shows the greatest measured tipping angle. The value of the angle can also be negative.
DYN Rotation (mm)
When measuring milling cutters with a rotating spindle, the control shows values in this area.
When measuring turning tools, the value DYN ROTATION describes the tolerance for the tipping angle. If the tolerance for the tipping angle is exceeded during calibration, the control marks the affected value in the MIN or MAX fields with an *.
In the optional machine parameter tippingTolerance (no. 114206) you define the tipping angle tolerance. The control will determine the tipping angle automatically only if a tolerance is defined.

</td></tr>
<tr><td>TT Tool cutting edges</td><td>Number
List of the measurements performed and the measured values of the individual cutting edges</td></tr>
</table>

Tool tab

On the **Tool** tab the control shows information about the active tool, depending on the tool type.

Further information: "Tool types", Page 281

Contents for dressing, milling, and grinding tools (option 156)

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Note on the tool
Tool geometry	■ **L** Tool length ■ **R** Tool radius ■ **R2** Corner radius of the tool
Tool allowances	■ **DL** Delta value for the tool length ■ **DR** Delta value for the tool radius ■ **DR2** Delta value for the corner radius of the tool With **Program**, the control displays the values from a tool call with **TOOL CALL** or from a tool compensation with a compensation table ***.tcs**. **Further information:** "Tool call", Page 304 **Further information:** "Tool compensation with compensation tables", Page 1120 With **Table**, the control displays the values from the tool management. **Further information:** "Tool management ", Page 297
Tool ages	■ **Cur. time (h:m)** Time in hours and minutes the tool has been engaged ■ **Time 1 (h:m)** Service life of the tool ■ **Time 2 (h:m)** Maximum service life at tool call
Replacement tool	■ **RT** Tool number of the replacement tool ■ **Name** Tool name of the replacement tool

Area	Contents
Tool type	■ **Tool Axis** Tool axis programmed in the tool call (e.g., **Z**) ■ **Type** Tool type of the active tool (e.g., **DRILL**)

Other contents for turning tools (option 50)

Area	Contents
Tool geometry	■ **ZL (mm)** Tool length in Z direction ■ **XL (mm)** Tool length in X direction ■ **RS (mm)** Cutter radius ■ **YL (mm)** Tool length in Y direction
Tool allowances	■ **DZL (mm)** Delta value in Z direction ■ **DXL (mm)** Delta value in X direction ■ **DRS (mm)** Delta value for the cutter radius ■ **DCW (mm)** Delta value for the width of the recessing tool
Tool type	■ **Tool Axis** ■ **TO** Tool orientation ■ **Type** Tool type, e.g. **TURN**

Definitions

OEM transformations for special turning kinematics
Machine manufacturers can define OEM transformations for special turning kinematics. Machine manufacturers need these transformations for milling-turning machines that have a different orientation than the tool coordinate system in the home position of their axes.

Tipping angle
If a TT tool touch probe with a cuboid contact cannot be clamped to a machine table so that it is level, the angular offset must be compensated for. This offset is the tipping angle.

Angle of misalignment
In order to exactly measure with TT tool touch probes with a cuboid contact, the misalignment on the machine table relative to the main axis must be compensated for. This offset is the angle of misalignment.

5.5 Simulation status workspace

Application

You can call additional status displays in the **Editor** operating mode in the **Simulation status** workspace. In the **Simulation status** workspace the control shows data based on the simulation of the NC program.

Description of function

The following tabs are available in the **Simulation status** workspace:

- **Favorites**
  Further information: "Favorites tab", Page 171
- **CYC**
  Further information: "CYC tab", Page 173
- **FN16**
  Further information: "FN16 tab", Page 173
- **LBL**
  Further information: "LBL tab", Page 175
- **M**
  Further information: "M tab", Page 176
- **PGM**
  Further information: "PGM tab", Page 178
- **POS**
  Further information: "POS tab", Page 179
- **QPARA**
  Further information: "QPARA tab", Page 180
- **Tables**
  Further information: "Tables tab", Page 181
- **TRANS**
  Further information: "TRANS tab", Page 181
- **TT**
  Further information: "TT tab", Page 183
- **Tool**
  Further information: "Tool tab", Page 184

5.6 Display of the program run time

Application

The control calculates the duration of all traverse movements and displays them together as the **Program run time**. The control takes traversing movements and dwell times into account.

In addition, the control calculates the remaining run time of the NC program.

Description of function

The control displays the program run time in the following areas:

- **PGM** tab of the **Status** workspace
- Status overview on the control bar
- **PGM** tab of the **Simulation status** workspace
- **Simulation** workspace in the **Editor** operating mode

Modify the **Settings** in the **Program run time** area in order to influence the calculated program run time.

Further information: "PGM tab", Page 178

The control opens a selection menu with the following functions:

Function	Meaning
Save	Save the current value under **Runtime**
Addition	Add the saved time to the value under **Runtime**
Resetting	Reset the saved time and the contents of the **Program run time** area to zero

The control counts the time during which the **Control-in-operation** symbol is green. The control adds the time from the **Program Run** operating mode and the **MDI** application.

The following functions reset the program run time:

- Selecting a new NC program for program run
- The **Reset program** button
- The **Resetting** function in the **Program run time** area

Remaining run time of the NC program

If a tool usage file is available, the control calculates for the **Program Run** operating mode how long the execution of the active NC program will take. During program run the control updates the remaining run time.

Further information: "Tool usage test", Page 312

The control shows the remaining run time in the status overview on the TNC bar.

The control does not take the feed-rate potentiometer setting into account, but calculates with a feed rate of 100%.

The remaining run time is reset by the following:

- Selecting a new NC program for program run
- The **Internal stop** button
- Generating a new tool usage file

Notes

- In the machine parameter **operatingTimeReset** (no. 200801) the machine manufacturer defines whether the control resets the program run time when the program is started.
- The control cannot simulate the run time of machine-specific functions such as tool changing. That is why this function is only partially suitable for calculating the production time in the **Simulation** workspace.
- In the **Program Run** operating mode, the control displays the exact time of the NC program while taking all machine-specific actions into account.

Definition

Control-in-operation:
The control uses the **Control-in-operation** symbol to show the machining status of the NC program or NC block:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

Further information: "Interrupting, stopping or canceling program run", Page 1959

When the control bar is expanded, the control shows additional information about the current status, such as **Active, feed rate at zero**.

5.7 Position displays

Application

The control offers various modes in the position display, for example values from different reference systems. You can choose one of the modes available based on the application.

Description of function

The control has position displays in the following areas:

- **Positions** workspace
- Status overview on the control bar
- **POS** tab of the **Status** workspace
- **POS** tab of the **Simulation status** workspace

On the **POS** tab of the **Simulation status** workspace the control always shows the **Nominal pos. (NOML)** mode. In the **Status** and **Positions** workspaces you can choose the mode of the position display.

The control offers the following modes for the position display:

Mode	Meaning
Nominal pos. (NOML)	This mode shows the value of the currently calculated target position in the input coordinate system **I-CS**. When the machine moves the axes, the control compares the coordinates of the measured actual position with the calculated nominal position in predefined time intervals. The nominal position is the position at which the axes should be located at the time of comparison, based on the calculation. The **Nominal pos. (NOML)** and **Actual pos. (ACT)** modes differ solely with regard to the servo lag.
Actual pos. (ACT)	This mode shows the currently measured tool position in the input coordinate system **I-CS**. The actual position is the measured position of the axes, as determined by encoders at the time of comparison.
Nominal reference position (RFNOML)	This mode shows the calculated target position in the machine coordinate system **M-CS**. The **Nominal reference position (RFNOML)** and **Actual reference position (RFACTL)** modes differ solely with regard to the servo lag.
Actual reference position (RFACTL)	This mode shows the currently measured tool position in the machine coordinate system **M-CS**.
Servo lag (LAG)	This mode shows the difference between the calculated nominal position and the measured actual position. The control determines the difference in predefined time intervals.
Handwheel superimposed (M118)	This mode shows the values that you move using the **M118** miscellaneous function. **Further information:** "Activating handwheel superimpositioning with M118", Page 1334

Refer to your machine manual.

In the machine parameter **progToolCallDL** (no. 124501) the machine manufacturer defines whether the position display takes the delta value **DL** from the tool call into account. The modes **NOML.** and **ACTL.** as well as **RFNOML** and **RFACTL** then differ from each other by the value **DL**.

5.7.1 Switching the position display mode

To switch the position display mode in the **Status** workspace:

- Select the **POS** tab
- Select **Settings** in the position display area
- Select the desired mode for the position display, for example **Actual pos. (ACT)**
- > The control displays the positions in the selected mode.

Notes

- The machine parameter **CfgPosDisplayPace** (no. 101000) defines the display accuracy by the number of decimal places.
- When the machine moves the axes, the control displays the distances-to-go of the individual axes with a symbol and the appropriate value next to the current position.

  Further information: "Axis display and position display", Page 164

5.8 Defining the contents of the QPARA tab

On the **QPARA** tab of the **Status** and **Simulation status** workspaces you can define which variables the control shows.

Further information: "QPARA tab", Page 180

To define the contents of the **QPARA** tab:

- Select the **QPARA** tab
- Select the **Settings** in the desired area, such as QL parameters
- > The control opens the **Parameter list** window.
- Enter numbers, such as **1,3,200-208**
- Press **OK**
- > The control displays the values of the defined variables.

- Use commas to separate single variables and connect sequential variables with a hyphen.
- The control always shows eight decimal places on the **QPARA** tab. For example, the control shows the result of **Q1 = COS 89.999** as 0.00001745. Very large and very small values are shown in exponential notation. The control shows the result of **Q1 = COS 89.999 * 0.001** as +1.74532925e-08, with e-08 corresponding to the factor of 10^{-8}.
- For variable texts in QS parameters the control shows the first 30 characters, i.e. the contents might be truncated.

6

Powering On and Off

6.1 Powering on

Application

After using the main switch to power on the machine, the control's boot process begins. The following steps may differ depending on the machine; for example, whether absolute or incremental position encoders are used.

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

Related topics

- Absolute and incremental position encoders
  Further information: "Position encoders and reference marks", Page 209

Description of function

⚠ 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

Power-on of the control begins with the power supply.

After booting, the controls checks the machine status, e.g.:

- Positions identical to before switching off the machine
- Safety features are ready, such as the emergency stop
- Functional safety

If the control registers an error during or after booting, it issues an error message.

The following step differs depending on position encoders on the machine:

- Absolute position encoders
  If the machine has absolute position encoders, the control is in the **Start/Login** application after power-on.
- Incremental position encoders
  If the machine has incremental position encoders, you must traverse the reference points in the **Move to ref. point** application. Once all axes have been referenced, the control is in the **Manual operation** application.
  Further information: "Referencing workspace", Page 197
  Further information: "Manual operation application", Page 202

6.1.1 Powering the machine and the control on

To switch the machine on:

- ▶ Switch the power supply of the control and of the machine on
- > The control is in start-up mode and shows the progress in the **Start/Login** workspace.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace.

OK

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

I

- ▶ Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit.
- > If the machine is equipped with absolute linear and angle encoders, the control is now ready for operation.
- > If the machine is equipped with incremental linear and angle encoders, the control opens the **Move to ref. point** application.

  Further information: "Referencing workspace", Page 197

- ▶ Press the **NC Start** key
- > The control moves to all necessary reference points.
- > The control is ready for operation and the **Manual operation** application is open.

  Further information: "Manual operation application", Page 202

Notes

NOTICE

Danger of collision!

When the machine is switched on, the control tries to restore the switch-off status of the tilted plane. This is prevented under certain conditions. For example, this applies if axis angles are used for tilting while the machine is configured with spatial angles, or if you have changed the kinematics.

- If possible, reset tilting before shutting the system down
- Check the tilted condition when switching the machine back on

NOTICE

Danger of collision!

Failure to notice deviations between the actual axis positions and those expected by the control (saved at shutdown) can lead to undesirable and unexpected axis movements. There is risk of collision during the reference run of further axes and all subsequent movements!

- Check the axis positions
- Only confirm the pop-up window with **YES** if the axis positions match
- Despite confirmation, at first only move the axis carefully
- If there are discrepancies or you have any doubts, contact your machine manufacturer

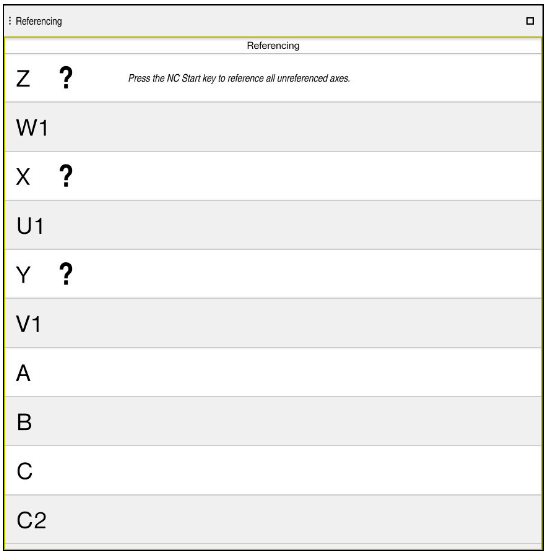
6.2 Referencing workspace

Application

On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.

Description of function

The **Referencing** workspace is always open in the **Move to ref. point** application. If reference points are to be traversed when powering-on the machine, then the control opens this application automatically.



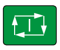
Referencing workspace with axes to be referenced

The control displays a question mark behind all axes that need to be referenced.

Once all axes have been referenced, the control closes the **Move to ref. point** application and switches to the **Manual operation** application.

6.2.1 Axis reference run

To reference the axes in the prescribed sequence:



- ▶ Press the **NC start** key
- > The control moves to the reference points.
- > The control switches to the **Manual operation** application.

To reference the axes in any sequence:

X+

- ▶ Press and hold the axis direction button for each axis until the reference point has been traversed
- > The control switches to the **Manual operation** application.

Notes

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision when referencing the axes.

- Pay attention to the information on the screen
- If necessary, move to a safe position before referencing the axes
- Watch out for possible collisions

- You cannot switch to the **Program Run** operating mode as long as reference points still need to be traversed.
- If you intend only to edit or simulate NC programs, you can switch to the **Editor** operating mode without referencing the axes. You can still traverse the reference points at a later time.

Notes about traversing reference points in a tilted working plane

If the **Tilt working plane** function (option 8) was active before the control was shut down, then the control will automatically reactivate this function after restarting. This means that movements via the axis keys take place in the tilted working plane.

Before traversing the reference points, you must deactivate the **Tilt working plane** function; otherwise, the control will interrupt the process with a warning. You can also home axes that are not activated in the current kinematic model without needing to deactivate **Tilt working plane**, such as a tool magazine.

Further information: "3-D rotation window (option 8)", Page 1098

6.3 Powering off

Application

To avoid losing data, shut down the control before powering-off the machine.

Description of function

You shut down the control in the **Start/Login** application of the **Home** operating mode.

If you select the **Shut down** button, the control opens the **Shut down** window. You choose whether to shut down the control or restart it.

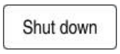
If NC programs or contours contain any unsaved changes, the control displays the unsaved changes in the **Close the program** window. You can save the changes, discard them or cancel the shutdown.

6.3.1 Shutting down the control and powering-off the machine

To power-off the machine:

- Select the **Home** operating mode

- Select **Shut down**
- > The control opens the **Shut down** window.



- Select **Shut down**
- > If NC programs or contours contain any unsaved changes, the control displays the **Close the program** window.
- If necessary, save unsaved NC programs with **Save** or **Save as**
- > The control shuts down.
- > Once shutdown has concluded, the control displays the text **Now you can switch off.**
- Switch off the main power switch of the machine

Notes

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss regardless of the control's status!

- Always shut down the control
- Only operate the main switch after being prompted on the screen

- Different machines have different power-off procedures.

  Refer to your machine manual.
- Applications on the control can delay the shutdown, such as a connection with **Remote Desktop Manager** (option 133)

  Further information: "Remote Desktop Manager window (option 133)", Page 2137

7

Manual Operation

7.1 Manual operation application

Application

In the **Manual operation** application you can manually move the axes and set up the machine.

Related topics

- Moving the machine axes
  Further information: "Moving the machine axes", Page 203
- Incremental jog positioning of machine axes
  Further information: "Incremental jog positioning of axes", Page 205

Description of function

The **Manual operation** application offers the following workspaces:

- **Positions**
- **Simulation**
- **Status**

The function bar in the **Manual operation** application contains the following buttons:

Button	Meaning
Handwheel	The control displays this toggle switch if a handwheel is configured for the control. If the handwheel is active, the operating mode's icon in the sidebar changes. **Further information:** "Electronic Handwheel", Page 2069
M	Define a miscellaneous function **M** or use the selection window to choose one and activate it with the **NC start** key. **Further information:** "Miscellaneous Functions", Page 1317
S	Define the spindle speed **S** activate it with the **NC start** key, and also switch on the spindle. **Further information:** "Spindle speed S", Page 309
F	Define the feed rate **F** and activate it with the **OK** button. **Further information:** "Feed rate F", Page 310
T	Define a tool **T** or use the selection window to choose one and insert it with the **NC start** key. **Further information:** "Tool call", Page 304
3D ROT	The control opens a window for the 3D rotation settings (option 8). **Further information:** "3-D rotation window (option 8)", Page 1098
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1366
DCM	The control opens the **Collision monitoring (DCM)** window, where you can activate or deactivate Dynamic Collision Monitoring (DCM, option 40). **Further information:** "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 1168

Button	Meaning
F limited	You activate or deactivate the feed-rate limitation for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 2096
Jog increment	Define the jog increment **Further information:** "Incremental jog positioning of axes", Page 205
Set the preset	Enter and set a preset **Further information:** "Preset management", Page 1025

Note

The machine manufacturer defines which miscellaneous functions are available on the control and which are allowed in the **Manual operation** application.

7.2 Moving the machine axes

Application

You can use the control to move the machine axes manually, such as pre-positioning for a manual touch probe function.

Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557

Related topics

- Programming traverse movements

  Further information: "Path Functions", Page 317
- Executing traverse movements in the **MDI** application

  Further information: "The MDI Application ", Page 1933

Description of function

The control offers the following methods for moving axes manually:

- Axis-direction keys
- Incremental jog positioning with the **Jog increment** button
- Traversing with electronic handwheels

  Further information: "Electronic Handwheel", Page 2069

The control displays the current contouring feed rate in the status display while the machine axes are in motion.

Further information: "Status Displays", Page 161

You can change the contouring feed rate with the **F** button in the **Manual operation** application and with the feed-rate potentiometer.

A traverse job is active on the control as soon as an axis moves. The control shows the status of the traverse job with the **Control-in-operation** icon in the status overview.

Further information: "Status overview on the TNC bar", Page 169

7.2.1 Using axis keys to move the axes

To move an axis manually with the axis keys:

- Select an operating mode (e.g., **Manual**)
- Select an application (e.g., **Manual operation**)

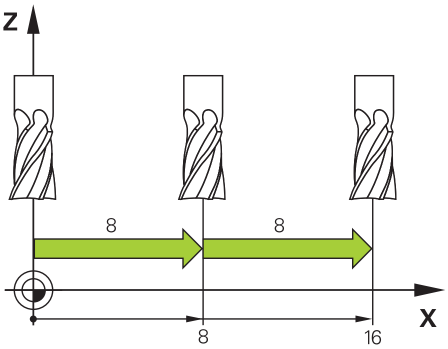
X+

- Press the axis key of the desired axis
- > The control moves the axis as long as you press the key.

> If you hold the axis key pressed down and simultaneously press the **NC start** key, the control moves the axis at a continuous feed rate. You have to end traverse movement with the **NC stop** key.
>
> You can move more than one axis at a time.

7.2.2 Incremental jog positioning of axes

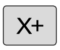
With incremental jog positioning you can move a machine axis by a preset distance. The input range for the infeed is from 0.001 mm to 10 mm.



To position an axis incrementally:

- Select the **Manual** operating mode
- Select the **Manual operation** application

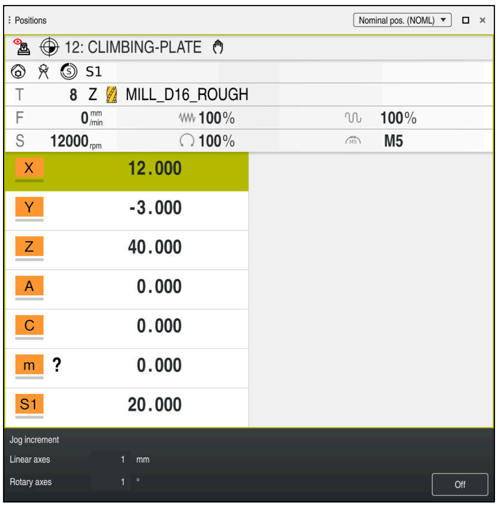
- Select **Jog increment**
- The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
- Enter the jog increment for linear axes and rotary axes



- Press the axis key of the desired axis
- The control positions the axis in the selected direction by the defined jog increment.

- Select **Jog increment On**
- The control ends incremental jog positioning and closes the **Jog increment** area in the **Positions** workspace.

> You can also end incremental jog positioning with the **Off** button in the **Jog increment** area.



Positions workspace with the **Jog increment** area active

Note

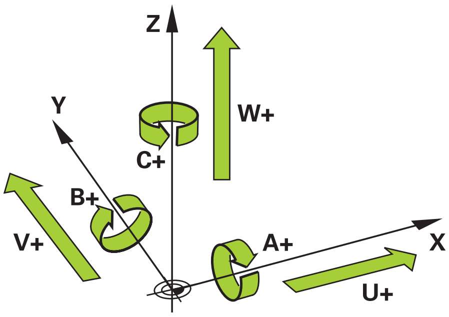
When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.

8

NC and Programming Fundamentals

8.1 NC fundamentals

8.1.1 Programmable axes



The programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

The programmable axes are designated as follows:

Main axis	Parallel axis	Rotary axis
X	U	A
Y	V	B
Z	W	C

Refer to your machine manual.

The number, designation and assignment of the programmable axes depend on the machine.

Your machine manufacturer can define further axes, such as PLC axes.

8.1.2 Designation of the axes on milling machines

The axes **X**, **Y** and **Z** on your milling machine are designated as the main axis (1st axis), secondary axis (2nd axis) and tool axis. The main axis and secondary axis define the working plane.

The axes are associated as follows:

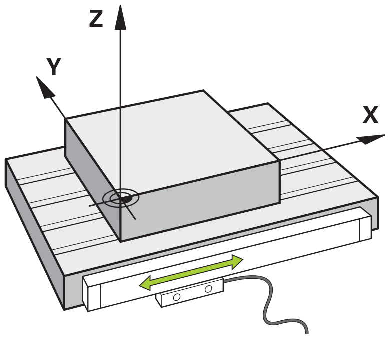
Main axis	Secondary axis	Tool axis	Working plane
X	Y	Z	**XY**, also **UV**, **XV**, **UY**
Y	Z	X	**YZ**, also **WU**, **ZU**, **WX**
Z	X	Y	**ZX**, also **VW**, **YW**, **VZ**

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.

8.1.3 Position encoders and reference marks

Fundamentals



The position of the machine axes is ascertained with position encoders. As a rule, linear axes are equipped with linear encoders. Rotary tables and rotary axes feature angle encoders.

The position encoders detect the positions of the tool or machine table by generating an electrical signal during movement of an axis. The control ascertains the position of the axis in the current reference system from this electrical signal.

Further information: "Reference systems", Page 1010

Position encoders can measure these positions through different methods:

- Absolutely
- Incrementally

The control cannot determine the position of the axes while the power is interrupted. Absolute and incremental position encoders behave differently once power is restored.

Absolute position encoders

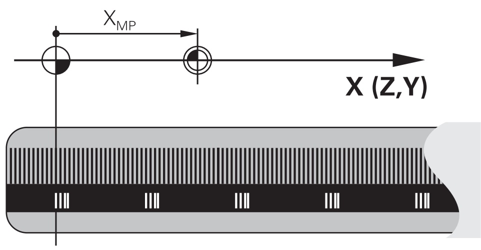
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

Incremental position encoders

Incremental position encoders need to find the distance between the current position and a reference mark in order to determine the actual position. Reference marks indicate a machine-based reference point. A reference mark must be traversed in order to determine the current position after a power interruption.

If the position encoders feature distance-coded reference marks, then you need to move the linear encoders of the axes by no more than 20 mm. On angle encoders this distance is no more than 20 °.

Further information: "Axis reference run", Page 197



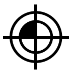
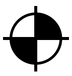
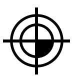
8.1.4 Presets in the machine

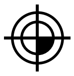
The following table contains an overview of the presets in the machine or on the workpiece.

Related topics

- Presets on the tool

  Further information: "Presets on the tool", Page 271

Icon	Preset
	**Machine datum** The machine datum is a fixed point defined in the machine configuration by the machine manufacturer. The machine datum is the origin of the machine coordinate system **M-CS**. **Further information:** "Machine coordinate system M-CS", Page 1012 If you program **M91** in an NC block, the defined values are referenced to the machine datum. **Further information:** "Traversing in the machine coordinate system M-CS with M91", Page 1322
 M92-ZP	**M92 datum M92-ZP (zero point)** The **M92** datum is a fixed point defined relative to the machine datum by the machine manufacturer in the machine configuration. The **M92** datum is the origin of the **M92** coordinate system. If you program **M92** in an NC block, the defined values are referenced to the **M92** datum. **Further information:** "Traversing in the M92 coordinate system with M92", Page 1323
	Tool change position The tool change position is a fixed point defined relative to the machine datum by the machine manufacturer in the tool-change macro.
	**Reference point** The reference point is a fixed point for initializing position encoders. **Further information:** "Position encoders and reference marks", Page 209 If the machine has incremental position encoders, the axes must traverse the reference point after booting. **Further information:** "Axis reference run", Page 197
	**Workpiece preset** With the workpiece preset you define the origin of the workpiece coordinate system **W-CS**. **Further information:** "Workpiece coordinate system W-CS", Page 1016 The workpiece preset is defined in the active row of the preset table. You determine the workpiece preset with a 3D touch probe, for example. **Further information:** "Preset management", Page 1025 If no transformations are defined, the entries in the NC program refer to the workpiece preset.

Icon	Preset
	**Workpiece datum** You define the workpiece datum with transformations in the NC program, for example with **TRANS DATUM** or a datum table. The entries in the NC program refer to the workpiece datum. If no transformations are defined in the NC program, the workpiece datum corresponds to the workpiece preset. If you tilt the working plane (option 8), the workpiece datum is the point around which the workpiece is rotated.

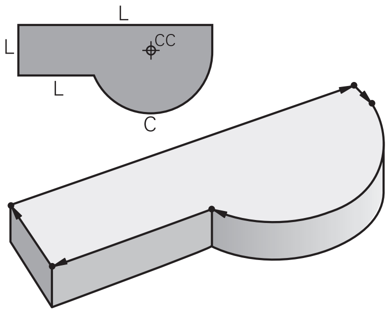
8.2 Programming possibilities

8.2.1 Path functions

Use the path functions to program contours.

A workpiece contour consists of several contour elements, such as straight lines and circular arcs. You use path functions, such as straight line **L**, to program tool movements for these contours.

Further information: "Fundamentals of path functions", Page 322



8.2.2 Graphical programming

As an alternative to Klartext programming you can program contours graphically in the **Contour graphics** workspace.

You can create 2D sketches by drawing lines and arcs and then export the contour to an NC program.

You can import existing contours from an NC program for graphical editing.

Further information: "Graphical Programming", Page 1437

8.2.3 Miscellaneous functions M

You can use miscellaneous functions to control the following actions:

- Program run (e.g., **M0** Program STOP)
- Machine functions (e.g., **M3** Spindle ON clockwise)
- Contouring behavior of the tool (e.g., **M197** Corner rounding)

Further information: "Miscellaneous Functions", Page 1317

8.2.4 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.

Program sections that are defined in a label can be directly executed repeatedly as program-section repeats, or can be called as a subprogram at defined locations in the main program.

If you wish to execute a specific NC program section only under certain conditions, you also define this machining sequence as a subprogram.

Within an NC program you can call a separate NC program for execution.

Further information: "Subprograms and program section repeats with the label LBL", Page 384

8.2.5 Programming with variables

In an NC program, variables are used as placeholders for numerical values or texts. A numerical value or text is assigned to a variable elsewhere.

In the **Q parameter list** window, you can see and edit the numerical values and texts of the individual variables.

Further information: "Q parameter list window", Page 1366

You can use the variables to program mathematical functions that control program execution or describe a contour.

You can also use variable programming, for example, to save and process measurement results determined by the 3D touch probe during program execution.

Further information: "Variables: Q, QL, QR and QS parameters", Page 1362

8.2.6 CAM programs

You can also optimize and execute externally created NC programs on the control.

You use CAD (**Computer-Aided Design**) to create geometric models of the workpieces to be produced.

In a CAM system (**Computer-Aided Manufacturing**) you then define how the CAD model will be produced. You can use an internal simulation to check the resulting tool paths, which are not control-specific.

With a postprocessor in the CAM system you then generate the control- and machine-specific NC programs. This results not only in programmable path functions, but also in splines (**SPL**) and straight lines **LN** with surface normal vectors.

Further information: "Multiple-Axis Machining", Page 1265

8.3 Programming fundamentals

8.3.1 Contents of an NC program

Application

You use NC programs to define the movements and behavior of your machine. NC programs consist of NC blocks that contain the syntax elements of the NC functions. With the HEIDENHAIN Klartext programming language, the control supports you by showing a dialog with information about the required content for every syntax element.

Related topics

- Creating a new NC program
  Further information: "Creating a new NC program", Page 132
- NC programs using CAD files
  Further information: "CAM-generated NC programs", Page 1301
- Structure of an NC program for contour machining
  Further information: "Structure of an NC program", Page 135

Description of function

You create NC programs in the **Editor** operating mode in the **Program** workspace.

Further information: "Program workspace", Page 217

The first and last NC blocks of the NC program contain the following information:

- Syntax **BEGIN PGM** or **END PGM**
- Name of the NC program
- Unit of measure of the NC program (mm or inches)

The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks when creating the NC program. You cannot delete these NC blocks.

The NC blocks created after **BEGIN PGM** contain the following information:

- Workpiece blank definition
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds
- Traverse movements, cycles and other NC functions

0 BEGIN PGM EXAMPLE MM	; Start of program
1 BLK FORM 0.1 Z X-50 Y-50 Z-20	; NC function for workpiece blank definition, consisting of two NC blocks
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL 5 Z S3200 F300	; NC function for tool call
4 L Z+100 R0 FMAX M3	; NC function for straight-line traverse
***** - ...**	
11 M30	; NC function for concluding the NC program
12 END PGM EXAMPLE MM	; End of program

Syntax component	Meaning
NC block	**4 TOOL CALL 5 Z S3200 F300** An NC block consists of the block number and the syntax of the NC function. An NC block can consist of multiple lines, such as with cycles. The control numbers the NC blocks in ascending sequence.
NC function	**TOOL CALL 5 Z S3200 F300** You use NC functions to define the behavior of the control. The block number is not a part of the NC functions.
Syntax initiator	**TOOL CALL** The syntax initiator clearly designates each NC function. Syntax initiators are used in the **Insert NC function** window. **Further information:** "Inserting NC functions", Page 228

Syntax component	Meaning
Syntax element	**TOOL CALL 5 Z S3200 F300** Syntax elements are all parts of the NC function, such as technology values **S3200** or coordinate information. NC functions also contain optional syntax elements. The control shows certain syntax elements in color in the **Program** workspace. **Further information:** "Appearance of the NC program", Page 219
Value	**3200** for spindle speed **S** Not every syntax element must contain a numerical value, such as tool axis **Z**.

If you create NC programs in a text editor or outside of the control, note the correct spelling and sequence of the syntax elements.

Notes

- NC functions can also consist of more than one NC block, such as **BLK FORM**.
- Miscellaneous functions **M** and comments can be both syntax elements within NC functions as well as their own NC functions.
- Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.
- The file name extension ***.h** designates a Klartext program.

  Further information: "Programming fundamentals", Page 212

8.3.2 Editor operating mode

Application

In the **Editor** operating mode you can do the following:

- Create, edit and simulate NC programs
- Create and edit contours
- Create and edit pallet tables

Description of function

With **Add** you can create a new file or open an existing one. The control displays up to ten tabs.

The **Editor** presents the following workspaces if an NC program is open:

- **Help**
  Further information: "Help workspace", Page 1506
- **Contour**
  Further information: "Graphical Programming", Page 1437
- **Program**
  Further information: "Program workspace", Page 217
- **Simulation**
  Further information: "Simulation Workspace", Page 1535
- **Simulation status**
  Further information: "Simulation status workspace", Page 186
- **Keyboard**
  Further information: "Virtual keyboard of the control bar", Page 1508

If you open a pallet table, the control displays the workspaces **Job list** and **Form** for pallets. You cannot edit these workspaces.

Further information: "Job list workspace", Page 1938

Further information: "Form workspace for pallets", Page 1946

If option 154 is active, **Batch Process Manager** lets you use the complete scope of functions when executing pallet tables.

Further information: "Job list workspace", Page 1938

If an NC program or pallet table is selected in the **Program Run** operating mode, the controls shows the **M** status on the tab of the NC program. If the **Simulation** workspace for this NC program is open, the controls shows the **Control-in-operation** icon on the tab of the NC program.

Icons and buttons

The **Editor** operating mode contains the following icons and buttons:

Icon or button	Meaning
	The control uses this icon to show that an NC program is open.
	The control uses this icon to show that a contour is open. **Further information:** "Graphical Programming", Page 1437
	The control uses this icon to show that a pallet table is open. **Further information:** "Pallet Machining and Job Lists", Page 1937
Klartext programming	If this toggle switch is active, then you are using dialog-guided programming. If this toggle switch is not active, then you are programming in the text editor. **Further information:** "Editing NC programs", Page 228
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Editing NC programs", Page 228
GOTO block number	The control selects the block number that you defined. **Further information:** "GOTO function", Page 1511
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1366
/ Skip block off/on	Hide NC blocks with a **/** character. NC blocks hidden with a **/** character will be ignored during program run as soon as the **Skip block** toggle switch is active. **Further information:** "Hiding NC blocks", Page 1513
; Comment Off/On	Insert or remove a **;** character in front of an NC block. If an NC block begins with a **;** character, then the block is a comment. **Further information:** "Adding comments", Page 1512
Edit	The control opens the context menu. **Further information:** "Context menu", Page 1522
Select in Program Run	The control opens the file in the **Program Run** operating mode. **Further information:** "Program Run", Page 1953
Start the simulation	The control opens the **Simulation** workspace and starts graphic simulation. **Further information:** "Simulation Workspace", Page 1535

8.3.3 Program workspace

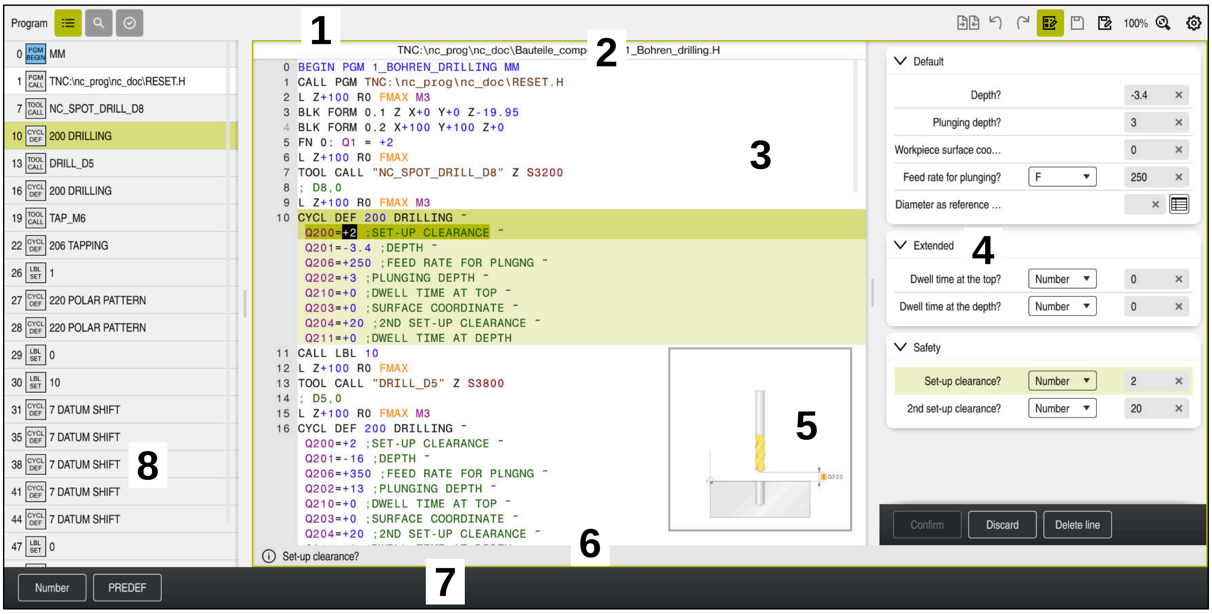
Application

The control displays the NC program in the **Program** workspace.

You can edit the NC program in the **Editor** operating mode and in the **MDI** application, but not in the **Program Run** operating mode.

Description of function

Areas of the Program workspace



Program workspace with active structure, help graphic and form

1 Title bar
  Further information: "Icons in the title bar", Page 219
2 File information bar
  In the file information bar the control shows the path and file name of the NC program. In the **Program Run** and **Editor** operating modes, the file information bar contains a breadcrumb navigation.
  Further information: "Navigation path in the Program workspace", Page 1962
3 Contents of the NC program
  Further information: "Appearance of the NC program", Page 219
4 **Form** column
  Further information: "Form column in the Program workspace", Page 227
5 Help graphic of the syntax element being edited
  Further information: "Help graphic", Page 220
6 Dialog bar
  In the dialog bar the control shows additional information or instructions for the syntax element being edited.
7 Action bar
  In the action bar the control shows selection possibilities for the syntax element being edited.
8 **Structure**, **Search** or **Tool check** column
  Further information: "Structure column in the Program workspace", Page 1514
  Further information: "Search column in the Program workspace", Page 1517
  Further information: "Tool usage test", Page 312

Icons in the title bar

The following icons are shown in the **Program** workspace in the title bar:

Further information: "Icons on the control's user interface", Page 123

Icon or shortcut	Function
	Open and close the **Structure** column **Further information:** "Structure column in the Program workspace", Page 1514
CTRL+F	Open and close the **Search** column **Further information:** "Search column in the Program workspace", Page 1517
	Open and close the **Tool check** column **Further information:** "Tool usage test", Page 312
	Activate and end comparison functions **Further information:** "Program comparison", Page 1520
	Open and close the **Form** column **Further information:** "Form column in the Program workspace", Page 227
100%	Font size of the NC program If you select the percent value, the control displays symbols for increasing and decreasing the font size.
	Set font size of the NC program to 100%
	Open the **Program settings** window **Further information:** "Settings in the Program workspace", Page 220

Appearance of the NC program

By default the control shows the syntax with black characters. The control displays the following syntax elements in color within the NC program:

Color	Syntax element
Brown	Text entries (e.g., tool name or file name)
Blue	■ Numerical values ■ Structure items and texts
Dark green	Comments
Purple	■ Variables ■ Miscellaneous functions **M**
Dark red	■ Definition of spindle speed ■ Definition of feed rate
Orange	Rapid traverse **FMAX**
Gray	■ Not to be executed **M1** miscellaneous function ■ Not to be executed NC block hidden with a **/** character

Help graphic

When you are editing an NC block, for some NC functions the control shows a help graphic for the current syntax element. The size of the help graphic depends on the size of the **Program** workspace.

The control shows the help graphic at the right edge of the workspace, or at the top or bottom edge. The help graphic is positioned in the half that does not contain the cursor.

When you tap or click the help graphic, the control maximizes the help graphic. If the **Help** workspace is open, the control displays the help graphic there.

Further information: "Help workspace", Page 1506

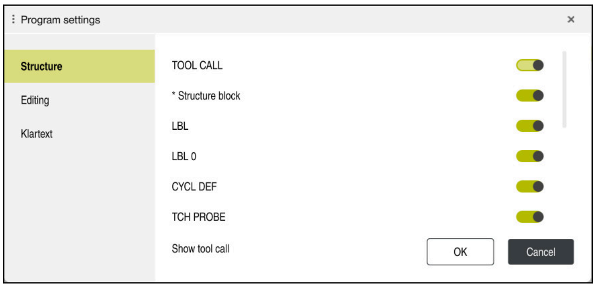
Settings in the Program workspace

In the **Program settings** window you can influence contents shown in the **Program** workspace as well as the control's behavior there. The selected settings are modally effective.

The settings available in the **Program settings** window depend on the operating mode or application. The **Program settings** window consists of the following areas:

Area	Editor operating mode	Program Run operating mode	MDI application
Structure	✓	✓	✓
Editing	✓	-	✓
Klartext	✓	-	✓
Tables	-	✓	-
FN 16	-	✓	-

Structure area



Structure area in the **Program settings** window

In the **Structure** area you use toggle switches to choose which structure elements the control should display in the **Structure** column.

Further information: "Structure column in the Program workspace", Page 1514

The following structure elements are available:

- **TOOL CALL**
- *** Structure block**
- **LBL**
- **LBL 0**
- **CYCL DEF**
- **TCH PROBE**
- **MONITORING SECTION START**
- **MONITORING SECTION STOP**
- **PGM CALL**
- **FUNCTION MODE**
- **M30 / M2**
- **M1**
- **M0 / STOP**
- **APPR / DEP**

Editing area

The **Editing** area contains the following settings:

Setting	Meaning
Automatic saving	**Save changes to the NC program automatically or manually** If the toggle switch is active, the control saves the NC program automatically upon the following actions: ■ Switching between tabs ■ Starting the simulation ■ Closing the NC program ■ Switching the operating mode If the toggle switch is not active, you must save manually. Upon the stated actions, the control asks whether the changes should be saved.
Allow syntax errors in text mode	If the toggle switch is active, the control can save NC blocks in the text editor even if they contain syntax errors. If the toggle switch is not active, you must correct all syntax errors within an NC block. Otherwise you cannot save the NC block. **Further information:** "Editing NC functions", Page 230
Generate absolute paths	**Create relative or absolute path entries** If the toggle switch is active, the control uses absolute paths for called files, e.g.: **TNC:\nc_prog\$mdi.h**. If the toggle switch is not active, the control uses relative paths , e.g.: **demo \reset.H**. If the file is located at a higher level in the folder structure than the calling NC program, the control creates an absolute path. **Further information:** "Path", Page 1148
Always save formatted	**Format NC program while saving** If an NC program has fewer than 30 000 characters, the control always formats the file when saving it, e.g.: capital letters for all syntax initiators. If the toggle switch is active, the control also formats NC programs with more than 30 000 characters each time it saves the file. This can increase the time needed for saving. If the toggle switch is not active, the control does not format NC programs with more than 30 000 characters.

Klartext area

In the **Klartext** area, select whether the control offers certain syntax elements of an NC block during input.

The control offers the following settings as toggle switches:

Setting	Meaning
Skip comment	If you activate this toggle switch, the control skips the comment function during programming for all NC functions. **Further information:** "Adding comments", Page 1512
Skip tool index	If you activate this toggle switch, the control skips the tool index for the following NC functions: ■ Call a tool with **TOOL CALL** **Further information:** "Tool call by TOOL CALL", Page 304 ■ Preselect a tool with **TOOL DEF** **Further information:** "Tool pre-selection by TOOL DEF", Page 311 **Further information:** "Indexed tool", Page 276
Skip linear superimposed interpolated axis values	If you activate this toggle switch, the control skips the **LIN_** syntax element for the following NC functions: ■ Circular contour **C** **Further information:** "Circular path C ", Page 334 ■ Circular contour **CR** **Further information:** "Circular path CR", Page 336 ■ Circular contour **CT** **Further information:** "Circular path CT", Page 339 **Further information:** "Linear superimpositioning of a circular path", Page 341

You can program the syntax elements in the form independently of the settings in the **Klartext** area.

Tables

In the **Tables** area you can select a unique table for each of the application areas shown, which is then active during program run.

Select the following tables using a selection window:

- **Datums**
  Further information: "Datum table", Page 2045
- **Tool correction**
  Further information: "Compensation table *.tco", Page 2055
- **Workpiece correction**
  Further information: "Compensation table *.wco", Page 2057

FN 16

In the **FN 16** area, use the **Show pop-up window** toggle switch to select whether the control displays a window in conjunction with **FN 16**.

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1382

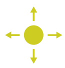
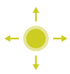
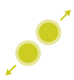
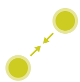
Using the Program workspace

The **Program** workspace can be used as follows:

- Touch operation
- Operation with keys and buttons
- Operation with a mouse

Touch operation

You use gestures to perform the following functions:

Symbol	Gesture	Meaning
	Tap	■ Select an NC block ■ Select a syntax element while editing
	Double tap	Edit an NC block
	Long press	Open the context menu If you are working with a mouse, click with the right mouse key. **Further information:** "Context menu", Page 1522
	Swipe	Scroll in an NC program
	Drag	Change the area in which NC blocks are marked. **Further information:** "Context menu in the Program workspace", Page 1525
	Spread	Increase the syntax font size
	Pinch	Reduce the syntax font size

Keys and buttons

You use keys and buttons to perform the following functions:

Key or button		Function
▲	▼	■ Navigate between NC blocks ■ During editing, search for the same syntax element in the NC program **Further information:** "Searching for the same syntax elements in different NC blocks", Page 226
▶	◀	■ Edit an NC block ■ During editing, navigate to previous or next syntax element
CTRL+ ▶	CTRL+ ◀	Navigate one position to the right or left within the value of a syntax element
GOTO		■ Use the block number to select an NC block directly **Further information:** "GOTO function", Page 1511 ■ Open selection menus during editing
⌖		Open position display of the control bar in order to copy the position If you select a line in the position display, the control copies the current value of this line to an open dialog.
CE		Delete value of a syntax element
NO ENT		Skip or remove optional syntax elements during programming
DEL BLK		Delete an NC block or cancel a dialog
END BLK		■ Confirm entry and conclude an NC block ■ Open the **Add** tab
ESC		Cancel editing without applying changes
Klartext editor		Select **Klartext programming** mode or a text editor **Further information:** "Editing NC functions", Page 230
Insert NC function		Open the **Insert NC function** window **Further information:** "Inserting NC functions", Page 228
Edit		Open the context menu **Further information:** "Context menu", Page 1522

Searching for the same syntax elements in different NC blocks

If you are editing an NC block, you can search for the same syntax element in the rest of the NC program.

To search for a syntax element in the NC program:

- Select an NC block
- Edit the NC block
- Navigate to the desired syntax element
- Press the arrow up or down key
- > The control marks the next NC block that contains the syntax element. The cursor is on the same syntax element as in the previous NC block. Press the arrow up key to search backwards.

Notes

- When you search for the same syntax element in a very long NC program, the control displays a pop-up window. You can cancel the search at any time.
- If the NC block contains a syntax error, the control precedes the block number with a corresponding icon. Click the icon to see the associated error description.
- Use the optional machine parameter **warningAtDEL** (no. 105407) to define whether the control shows a confirmation prompt in a pop-up window before an NC block is deleted.
- Use the machine parameter **stdTNChelp** (no. 105405) to define whether the control displays help graphics as pop-up windows in the **Program** workspace.

  When the **Help** workspace is open, the control displays the help graphic there, independent of the setting of this machine parameter.

  Further information: "Help workspace", Page 1506
- Use the optional machine parameter **maxLineCommandSrch** (no. 105412) to define how many NC blocks the control searches for the same syntax element.
- When you open an NC program, the control checks whether the NC program is complete and syntactically correct.

  Use the optional machine parameter **maxLineGeoSrch** (no. 105408) to define up to which NC block the control should check the program.
- If you open an NC program without content, you can edit the **BEGIN PGM** and **END PGM** NC blocks and change the unit of measure of the NC program.
- An NC program is incomplete without the **END PGM** NC block.

  If you open an incomplete NC program in the **Editor** operating mode, the control automatically adds this NC block.
- You cannot edit an NC program in the **Editor** operating mode if this NC program is currently being executed in the **Program Run** operating mode.

Form column in the Program workspace

Application

In the **Form** column in the **Program** workspace, the control shows all possible syntax elements for the currently selected NC function. You can edit all syntax elements in the form.

Related topics

- **Form** workspace for pallet tables
  Further information: "Form workspace for pallets", Page 1946
- Editing an NC function in the **Form** column
  Further information: "Editing NC functions", Page 230

Requirement

- **Klartext programming** mode must be active

Description of function

The control offers the following icons and buttons for using the **Form** column:

Icon or button	Function
	Show and hide the **Form** column
Confirm	Confirm entry and conclude an NC block
Discard	Discard entries and conclude an NC block
Delete line	Delete NC block

The control groups the syntax elements in the form depending on their functions, such as coordinates or safety.

The control indicates the required syntax elements with a red frame. Only once you have defined all of the required syntax elements can you confirm the entries and conclude the NC block. The control highlights the syntax element currently being edited.

If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

Notes

- In the following cases the control shows no contents in the form:
  - NC program is being run
  - NC blocks are being marked
  - NC block contains syntax error(s)
  - **BEGIN PGM** or **END PGM** NC blocks are selected
- If you define more than one miscellaneous function in an NC block, you can use the arrows in the form to change the sequence of the miscellaneous functions.
- If you define a label with a number, the control shows an icon next to the input area. The control uses this symbol to assign the next available number to the label.

8.3.4 Editing NC programs

Application

The editing of NC programs refers both to the insertion of NC functions as well as their modification. You can also edit NC programs that you previously generated with a CAM system and then transmitted to the control.

Related topics

- Using the **Program** workspace

  Further information: "Using the Program workspace", Page 224

Requirements

You can edit NC programs only in the **Editor** operating mode and in the **MDI** application.

> In the **MDI** application you edit only the NC program **$mdi.h** or **$mdi_inch.h**.

Description of function

Inserting NC functions

Inserting an NC function directly with keys or buttons

You can directly insert frequently needed NC functions, such as path functions, with keys.

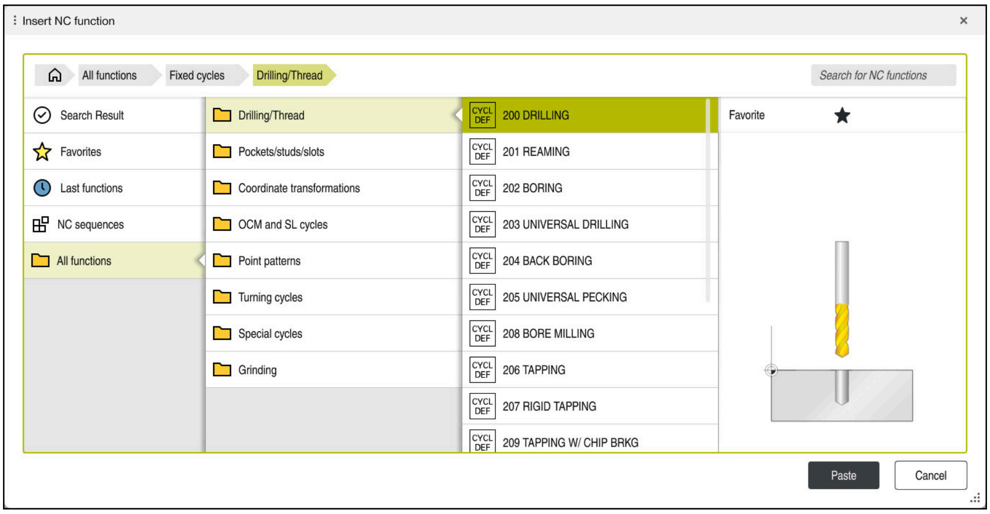
As an alternative to the keys, the control offers both the screen keyboard as well as the **Keyboard** workspace in NC input mode.

Further information: "Virtual keyboard of the control bar", Page 1508

To insert frequently needed NC functions:

- ▶ Select **L**
- > The control creates a new NC block and starts the dialog.
- ▶ Follow the instructions in the dialog

Inserting an NC function through selection



Insert NC function window

You can select all NC functions through the **Insert NC function** window.

You can navigate through the **Insert NC function** window as follows:

- Navigate through the tree structure manually, starting from **All functions**
- Use keys or buttons to narrow down the selection possibilities (e.g., **CYCL DEF** key opens the cycle groups)

  Further information: "Keycaps for NC dialog", Page 119
- Ten most recently used NC functions under **Last functions**
- NC functions marked as favorites under **Favorites**

  Further information: "Icons on the control's user interface", Page 123
- Stored sequence of NC functions under **NC sequences**

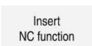
  Further information: "NC sequences for reuse", Page 392
- Enter a search term under **Search for NC functions**

  The control displays the results under **Search result**.

> You can begin the search as soon as the **Insert NC function** window opens by entering a character.

In the areas **Search result**, **Favorites** and **Last functions**, the control shows the path of the NC functions.

To insert a new NC function:



- Select **Insert NC function**
- The control opens the **Insert NC function** window.
- Navigate to the desired NC function
- The control highlights the selected NC function.

- Select **Paste**
- The control creates a new NC block and starts the dialog.
- Follow the instructions in the dialog

Inserting an NC function in the text editor

In the text editor, the control provides an auto-complete function when programming.

> If text editor mode is active, the **Klartext programming** toggle switch is to the left and gray.

To insert an NC function:

- Press the enter key
- > The control inserts an NC block.
- Enter the first letter of the NC function as needed
- Press the keyboard shortcut **CTRL+BLANK**
- > The control shows a selection menu with possible syntax initiators.
- Select the syntax initiator
- Enter the value as needed
- Press the keyboard shortcut **CTRL+BLANK** again if needed
- Select the syntax element as needed

> - If you press **CTRL+BLANK** immediately after entering a character string, the control displays a selection menu for the current syntax element.
> - If you insert a space character after a completely entered syntax element and then press **CTRL+BLANK**, the control displays a selection menu for the subsequent syntax element.

Editing NC functions

Editing an NC function in the Klartext programming mode

By default, the control opens newly created and syntactically correct NC programs in the **Klartext programming** mode.

To edit an existing NC function in the **Klartext programming** mode:

- Navigate to the desired NC function
- Navigate to the desired syntax element
- > The control displays alternative syntax elements in the action bar.
- Select a syntax element
- Define a value, if necessary

END BLK

- Conclude entry (e.g., by pressing **END**)

Editing an NC function in the Form column

If the **Klartext programming** mode is active, you can also use the **Form** column.

The **Form** column does not just show the syntax elements selected and used, but rather all those that are possible for the current NC function.

To edit an existing NC function in the **Form** column:

- Navigate to the desired NC function
- Show the **Form** column
- Select an alternative syntax element if necessary (e.g., **LP** instead of **L**)
- If necessary, edit or add the value
- If necessary, enter an optional syntax element or select from a list (e.g., miscellaneous function **M8**)

Confirm

- Conclude entry (e.g., by pressing **Confirm**)

Editing an NC function in the text editor mode

The control tries to correct syntax errors in the NC program automatically. If automatic correction is not possible, the control switches to text editor mode while editing this NC block. You must correct all errors before you can switch to **Klartext programming** mode.

- If text editor mode is active, the **Klartext programming** toggle switch is to the left and gray.
- If you are editing an NC block with syntax errors, the only way to cancel editing is to press the **ESC** key.

To edit an existing NC function in the text editor mode

> The control underscores the faulty syntax element with a jagged red line and shows an information symbol before the NC function (e.g., for **FMX** instead of **FMAX**).

- Navigate to the desired NC function
- Select the information symbol as needed

> The control displays the corresponding error description.

- Close the NC block

> The control might open the **NC block auto-correction** window with a solution proposal.

Yes

- Apply the proposal to the NC program with **Yes** or cancel auto-correction.

- The control cannot offer solution proposals in all cases.
- The text editor mode supports all navigation possibilities of the **Program** workspace. But you can work more quickly in the text editor mode by using gestures or a mouse, since then you can select the information symbol directly, for example.

Notes

- The instructions include emphasized text strings (e.g., **200 DRILLING**. You can use these text strings for better searching in the **Insert NC function** window.
- When you are editing an NC function, use the arrows to navigate left and right to the syntax elements, even within cycles. The up and down arrows search for the same syntax element in the rest of the NC program.

  Further information: "Searching for the same syntax elements in different NC blocks", Page 226
- If you are editing an NC block and haven't saved yet, the **Undo** and **Redo** functions affect the individual syntax elements of the NC function.

  Further information: "Icons on the control's user interface", Page 123
- Press the **actual position capture** key for the control to open the position display of the status overview. You can copy the current value of an axis into the programming dialog.

  Further information: "Status overview on the TNC bar", Page 169
- Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.
- You cannot edit an NC program in the **Editor** operating mode if this NC program is currently being executed in the **Program Run** operating mode.
- If you select an NC function in the **Insert NC function** window and swipe to the right, the control displays the following file functions:
  - Add to or remove from favorites
  - Navigate to the NC function

    Not in the **All functions** area
- In the areas **Search result**, **Favorites** and **Last functions**, the control shows the path of the NC functions.
- If software options are not enabled, the control shows unavailable contents in the **Insert NC function** window grayed out.

9

Technology-Specific NC Programming

9.1 Switching the operating mode with FUNCTION MODE

Application

The control offers a **FUNCTION MODE** operating mode for each of the technologies milling, milling-turning and grinding. Additionally, you can use **FUNCTION MODE SET** to activate settings defined by the machine manufacturer (e.g., switching the traverse range).

Related topics

- Milling-turning operations (option 50)

  Further information: "Turning (option 50)", Page 236
- Grinding operations (option 156)

  Further information: "Grinding operations (option 156)", Page 248
- Editing kinematic models in the **Settings** application

  Further information: "Channel settings", Page 2104

Requirements

- Control adapted by the machine manufacturer

  The machine manufacturer defines which internal functions the control performs with this function. The machine manufacturer must define selection possibilities for the **FUNCTION MODE SET** function.
- For **FUNCTION MODE TURN**, Milling/turning (software option 50)
- For **FUNCTION MODE GRIND**, Jig Grinding (software option 156)

Description of function

When the operating modes are switched, the control executes a macro that defines the machine-specific settings for the specific operating mode. With the NC functions **FUNCTION MODE TURN** and **FUNCTION MODE MILL**, you can activate a machine kinematic model that the machine manufacturer has defined and saved in the macro.

If the machine manufacturer has enabled the selection of various kinematic models, then you can switch between them using the **FUNCTION MODE** function.

If turning mode is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 163

Input

12 FUNCTION MODE TURN "AC_TURN"	; Activate turning mode with the selected kinematic model
11 FUNCTION MODE SET "Range1"	; Activate the machine manufacturer setting

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION MODE	Syntax initiator for the machining mode
MILL, **TURN**, **GRIND** or **SET**	Select the machining mode or machine manufacturer setting
" " or **QS**	Name of a kinematic model or machine manufacturer setting or QS parameter with the name You use choose the setting from a selection menu. Optional syntax element

Notes

WARNING

Caution: Danger to the operator and machine!

Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

- Clamp the workpiece in the spindle center
- Clamp workpiece securely
- Program low spindle speeds (increase as required)
- Limit the spindle speed (increase as required)
- Eliminate unbalance (calibrate)

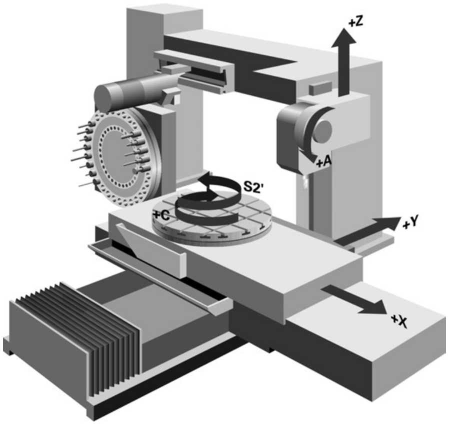
- In the optional machine parameter **CfgModeSelect** (no. 132200), the machine manufacturer defines the settings for the **FUNCTION MODE SET** function. If the machine manufacturer does not define the machine parameter, then **FUNCTION MODE SET** is not available.
- If the **Tilt working plane** or **TCPM** functions are active, you cannot switch the operating mode.
- The preset must be in the center of the turning spindle in turning mode.

9.2 Turning (option 50)

9.2.1 Fundamentals

Depending on the machine and kinematics, it is possible to perform both milling and turning operations on milling machines. A workpiece can thus be machined completely on one machine, even if complex milling and turning applications are required.

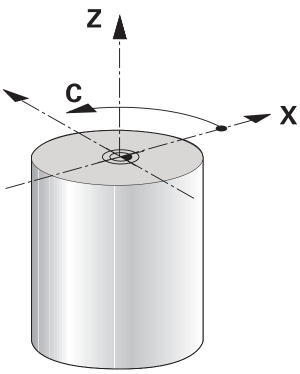
In a turning operation, the tool is in a fixed position, whereas the rotary table and the clamped workpiece rotate.



NC fundamentals for turning

The assignment of the axes with turning is defined so that the X coordinates describe the diameter of the workpiece and the Z coordinates the longitudinal positions.

Machining is thus always done in the **ZX** working plane. The machine axes to be used for the required movements depend on the respective machine kinematics and are determined by the machine manufacturer. This makes NC programs with turning functions largely exchangeable and independent of the machine model.



Workpiece preset for turning operations

On the control, you can simply switch between milling and turning mode within your NC program. In turning mode, the rotary table serves as lathe spindle, whereas the milling spindle with the tool is fixed. This way, it is possible to machine rotationally symmetric contours. The tool reference point must always be at the center of the lathe spindle.

Further information: "Preset management", Page 1025

If you use a facing slide, you can set the workpiece preset to a different location, since in this case the tool spindle performs the turning operation.

Further information: "Using a facing slide with FACING HEAD POS (option 50)", Page 1291

Production processes

Depending on the machining direction and task, turning applications can be subdivided into different production processes, e.g.:

- Longitudinal turning
- Face turning
- Recess turning
- Thread cutting

The control provides several cycles for each of the various production processes.

Further information: "Cycles for milling and turning", Page 744

You can run the cycles with an inclined tool in order to produce undercuts.

Further information: "Inclined turning", Page 240

Tools for turning operations

When managing turning tools, other geometric descriptions than those for milling or drilling tools are required. To execute a tool-tip radius compensation, for example, the definition of the cutting-edge radius is required. The control provides a special tool table for turning tools. In tool management, the control displays only the required tool data for the current tool type.

Further information: "Tool data", Page 275

Further information: "Tooth radius compensation for turning tools (option 50)", Page 1117

You can correct turning tool values in the NC program.

The control offers the following functions for this:

- Cutter radius compensation

  Further information: "Tooth radius compensation for turning tools (option 50)", Page 1117
- Compensation tables

  Further information: "Tool compensation with compensation tables", Page 1120
- The **FUNCTION TURNDATA CORR** function

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1124

Notes

⚠ WARNING

Caution: Danger to the operator and machine!

Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

- Clamp the workpiece in the spindle center
- Clamp workpiece securely
- Program low spindle speeds (increase as required)
- Limit the spindle speed (increase as required)
- Eliminate unbalance (calibrate)

- The orientation of the tool spindle (spindle angle) depends on the machining direction. The tool tip is aligned to the center of the turning spindle for outside machining. For inside machining, the tool points away from the center of the turning spindle.

  The direction of spindle rotation must be adapted when the machining direction (outside/inside machining) is changed.

  Further information: "Overview of miscellaneous functions", Page 1319
- During turning, the cutting edge and the center of the turning spindle must be at the same level. During turning, the tool therefore has to be pre-positioned to the Y coordinate of the turning-spindle center.
- In turning mode, diameter values are displayed on the X axis position display. The control then shows an additional diameter symbol.

  Further information: "Positions workspace", Page 163
- In turning mode, the spindle potentiometer is active for the turning spindle (rotary table).
- In turning mode, no coordinate conversion cycles are permitted except for the datum shift.

  Further information: "Datum shift with TRANS DATUM", Page 1046
- In turning mode, the **SPA**, **SPB** and **SPC** transformations from the preset table are not permitted. If you activate one of these transformations while executing the NC program in turning mode, the control will display the **Transformation not possible** error message.
- The machining times determined using the graphic simulation do not correspond to the actual machining times. Reasons for this during combined milling-turning operations include the switching of operating modes.

  Further information: "Simulation Workspace", Page 1535

9.2.2 Technology values for turning operations

Defining the spindle speed for turning with FUNCTION TURNDATA SPIN

Application

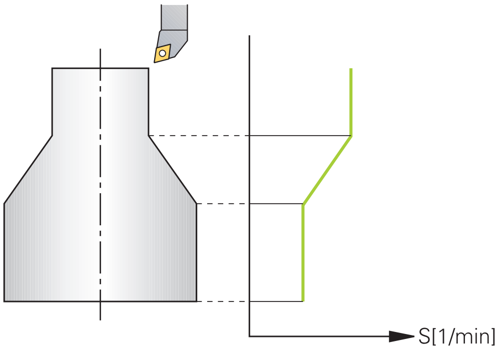
With turning you can machine both at constant spindle speed and constant cutting speed.

Use **FUNCTION TURNDATA SPIN** to define the speed.

Requirement

- Machine with at least two rotary axes
- Combined milling/turning (software option 50)

Description of function



If you machine at constant cutting speed **VCONST:ON**, the control modifies the speed according to the distance of the tool tip to the center of the turning spindle. For positioning movements toward the center of rotation, the control increases the table speed; for movements away from the center of rotation, it reduces the table speed.

For processing with constant spindle speed **VCONST:Off**, speed is independent of the tool position.

With **FUNCTION TURNDATA SPIN** you can define a maximum speed for the constant speed.

Input

11 FUNCTION TURNDATA SPIN VCONST:ON VC:100 GEARRANGE:2	; Constant surface speed with gear range 2

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA SPIN	Syntax initiator for speed definition in turning mode
VCONST OFF or **ON**	Definition of a constant cutting speed or constant surface speed Optional syntax element
VC	Value for the surface speed Optional syntax element
S or **SMAX**	Constant speed or speed limitation Optional syntax element
GEARRANGE	Gear range for the lathe spindle Optional syntax element

Notes

- If you machine at constant cutting speed, the selected gear range limits the possible spindle speed range. The possible gear ranges (if applicable) depend on your machine.
- When the maximum speed has been reached, the control displays **SMAX** instead of **S** in the status display.
- To reset the speed limitation, program **FUNCTION TURNDATA SPIN SMAX0**.
- In turning mode, the spindle potentiometer is active for the turning spindle (rotary table).
- Cycle **800** limits the maximum spindle speed during eccentric turning. The control restores a programmed limitation of the spindle speed after eccentric turning.

  Further information: "Cycle 800 ADJUST XZ SYSTEM ", Page 748

Feed rate

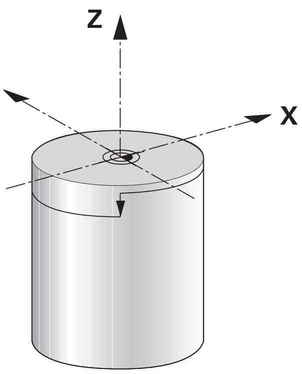
Application

With turning, feed rates are often specified in millimeters per revolution. Use the miscellaneous function **M136** for this on the control.

Further information: "Interpreting the feed rate as mm/rev with M136", Page 1346

Description of function

With turning, feed rates are often specified in millimeters per revolution. The control thus moves the tool at a defined value for every spindle rotation. The resulting contouring feed rate is thus dependent on the speed of the turning spindle. The control increases the feed rate at high spindle speeds and reduces it at low spindle speeds. This enables you to machine with uniform cutting depth and constant cutting force, thus achieving constant chip thickness



Note

During many turning operations, it is not possible to maintain constant surface speeds (**VCONST: ON**) because the maximum spindle speed is reached first. Use the machine parameter **facMinFeedTurnSMAX** (no. 201009) to define the behavior of the control after the maximum speed has been reached.

9.2.3 Inclined turning

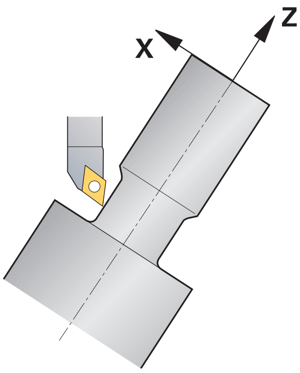
Application

It may sometimes be necessary for you to bring the swivel axes into a specific position to machine a specific process. This can be necessary, for example, when you can only machine contour elements according to a specific position due to tool geometry.

Requirement

- Machine with at least two rotary axes
- Combined milling/turning (software option 50)

Description of function



The control offers the following methods of inclined turning:

Function	Description	Further information
M144	The control uses **M144** in subsequent traverse movements to compensate for tool offsets that result from inclined rotary axes.	Page 1350
M128	With **M128** the control behaves like with **M144**, but you cannot use cutter radius compensation outside of cycles.	Page 1341
FUNCTION TCPM with **REFPNT TIP-CENTER**	Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip. If you activate inclined machining with **FUNCTION TCPM** with **REFPNT TIP-CENTER**, then tool-tip radius compensation is also possible without a cycle; that is, in traversing blocks with **RL**/**RR**. HEIDENHAIN recommends using **FUNCTION TCPM** with **REFPNT TIP-CENTER**.	Page 1104
Cycle **800**	Use Cycle **800 ADJUST XZ SYSTEM** to define an inclination angle.	Page 748

If you execute turning cycles with **M144**, **FUNCTION TCPM**, or **M128**, then the angles of the tool relative to the contour will change. The control automatically takes these modifications into account and therefore also monitors the inclined machining operation.

Notes

- Threading cycles can be run with inclined machining only if the tool is at a right angle (+90°, or –90°).
- Tool compensation **FUNCTION TURNDATA CORR-TCS** is always active in the tool coordinate system, even during inclined machining.

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1124

9.2.4 Simultaneous turning

Application

You can combine the turning operation with function **M128** or **FUNCTION TCPM** and **REFPNT TIP-CENTER**. This enables you to manufacture contours in one cut, for which you have to change the inclination angle (simultaneous machining).

Related topics

- Cycles for simultaneous turning (option 158)

  Further information: "Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option158) ", Page 887
- Miscellaneous function **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1341
- **FUNCTION TCPM** (option 9)

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

Requirements

- Machine with at least two rotary axes
- Combined milling/turning (software option 50)
- Advanced Functions Set 2 (software option 9)

Description of function

The simultaneous turning contour is a turning contour for which a rotary axis whose inclination does not violate the contour can be programmed on **CP** polar circles and **L** linear blocks. Collisions with lateral cutting edges or holders are not prevented. This makes it possible to finish contours with one tool in a continuous movement, even though different sections of the contour are accessible only in different tool inclinations.

In the NC program you define how the rotary axis has to be inclined to reach the different contour parts without collisions.

Use the cutter radius oversize **DRS** to leave an equidistant oversize on the contour.

Use **FUNCTION TCPM** and **REFPNT TIP-CENTER** to measure the theoretical tool tip of the turning tools being used for this.

The following requirements apply if you want to use **M128** for simultaneous turning:

- Only for NC programs programmed on the path of the tool center.
- Only for button turning tools with TO 9

  Further information: "Subgroups of technology-specific tool types", Page 282
- The tool must be measured at the center of the tool-tip radius

Further information: "Presets on the tool", Page 271

Example

An NC program with simultaneous turning includes the following components:

- Activate turning mode
- Insert a turning tool
- Adjust the coordinate system with cycle **800 ADJUST XZ SYSTEM**
- Activate **FUNCTION TCPM** with **REFPNT TIP-CENTER**
- Activate cutter radius compensation with **RL/RR**
- Program simultaneous turning contour
- End cutter radius compensation with **R0** or by departing the contour
- Reset **FUNCTION TCPM**

0 BEGIN PGM TURNSIMULTAN MM	
*** - ...**	
12 FUNCTION MODE TURN	; Activate turning mode
13 TOOL CALL "TURN_FINISH"	; Insert turning tool
14 FUNCTION TURNDATA SPIN VCONST:OFF S500	
15 M140 MB MAX	
*** - ...**	; Adjust the coordinate system
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+90 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
17 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	; Activate **FUNCTION TCPM**
18 FUNCTION TURNDATA CORR-TCS:Z/X DRS:-0.1	
19 L X+100 Y+0 Z+10 R0 FMAX M304	
20 L X+45 RR FMAX	; Activate cutter radius compensation with **RR**
*** - ...**	
26 L Z-12.5 A-75	; Program simultaneous turning contour
27 L Z-15	
28 CC X+69 Z-20	
29 CP PA-90 A-45 DR-	
30 CP PA-180 A+0 DR-	
*** - ...**	
47 L X+100 Z-45 R0 FMAX	; End cutter radius compensation with **R0**
48 FUNCTION RESET TCPM	; Reset **FUNCTION TCPM**
49 FUNCTION MODE MILL	
*** - ...**	
71 END PGM TURNSIMULTAN MM	

9.2.5 Turning operation with FreeTurn tools

Application

The control makes it possible to define FreeTurn tools and to use them, for example, for inclined or simultaneous turning operations.

FreeTurn tools are lathe tools that are equipped with multiple cutting edges. Depending on the variant, a single FreeTurn tool may be capable of axis-parallel and contour-parallel roughing and finishing.

Thanks to the use of FreeTurn tools, fewer tool changes are required, reducing the machining time. Due to the tool orientation to the workpiece, only outside machining is possible.

Related topics

- Inclined turning

  Further information: "Inclined turning", Page 240
- Simultaneous turning operation

  Further information: "Simultaneous turning", Page 242
- FreeTurn tools

  Further information: "Tool data", Page 275
- Indexed tools

  Further information: "Indexed tool", Page 276

Requirements

- Machine whose tool spindle is perpendicular to the workpiece spindle or can be inclined.

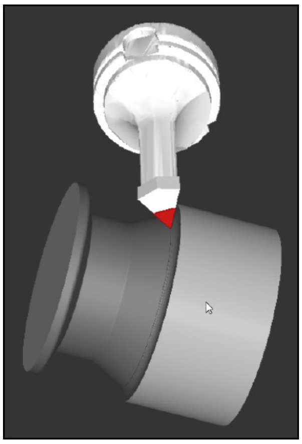
  Depending on the machine kinematics, a rotary axis is required for the orientation of the spindles to each other.
- Machine with controlled tool spindle

  The control inclines the cutting edge by means of inclining the tool spindle.
- Combined milling/turning (software option 50)
- Kinematics description

  The machine manufacturer provides the kinematics description. Based on the kinematics description, the control can take the tool geometry, for example, into account.
- Machine-manufacturer macros for simultaneous turning with FreeTurn tools
- FreeTurn tool with suitable tool carrier
- Tool definition

  A FreeTurn tool always includes three cutting edges of an indexed tool.

Description of function

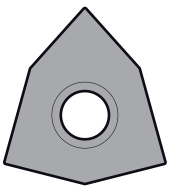


FreeTurn tool in simulation

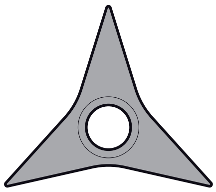
To use FreeTurn tools, call only the desired cutting edge of the correctly defined indexed tool in your NC program.

Further information: "Example: Turning with a FreeTurn tool", Page 906

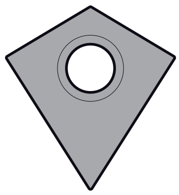
FreeTurn tools



FreeTurn indexable insert for roughing



FreeTurn indexable insert for finishing



FreeTurn indexable insert for roughing and finishing

The control supports all variants of FreeTurn tools:

- Tool with finishing cutting edge
- Tool with roughing cutting edge
- Tool with finishing and roughing cutting edge

In the **TYP** column of the tool management, select a turning tool (**TURN**) as the tool type. In the **TYPE** column, assign the appropriate technology-specific tool type to each cutting edge, i.e. roughing tool (**ROUGH**) or finishing tool (**FINISH**).

Further information: "Subgroups of technology-specific tool types", Page 282

A FreeTurn tool must be defined as an indexed tool with three cutting edges that are offset by the **ORI** angle of orientation. Each cutting edge has the **TO 18** tool orientation.

Further information: "Example of a FreeTurn tool", Page 280

FreeTurn tool carrier

Tool carrier template for a FreeTurn tool

There is a suitable tool carrier for each FreeTurn tool variant. HEIDENHAIN provides ready-to-use tool carrier templates for download that are included in the programming station software. You can then assign the tool-carrier kinematics descriptions generated from the templates to the respective indexed cutting edge.

Further information: "Tool-carrier templates", Page 302

Notes

NOTICE
Danger of collision! The shaft length of the turning tool limits the diameter that can be machined. There is a risk of collision during machining! ▶ Check the machining sequence in the simulation

- Due to the tool orientation to the workpiece, only outside machining is possible.
- Please note that FreeTurn tools can be combined with various machining strategies. Therefore, make sure to observe the specific notes, e.g. in conjunction with the selected machining cycles.

9.2.6 Unbalance in turning operations

Application

In a turning operation, the tool is in a fixed position, whereas the rotary table and the clamped workpiece rotate. Depending on the size of the workpiece, the mass that is set in rotation can be very large. As the workpiece rotates, it creates an outward centrifugal force.

The control offers functions to detect the unbalance and support you in compensating for it.

Related topics

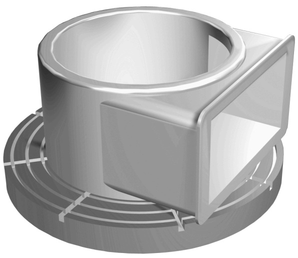
- Cycle **892 CHECK UNBALANCE**
  Further information: "Cycle 892 CHECK UNBALANCE ", Page 757
- Cycle **239 ASCERTAIN THE LOAD** (option 143)
  Further information: "Cycle 239 ASCERTAIN THE LOAD (option 143)", Page 1232

Description of function

Refer to your machine manual.

Unbalance functions are not required and available on all machine tool types.

The unbalance functions described here are basic functions that are set up and adapted to the machine by the machine manufacturer. The scope and effect of the described functions may therefore vary from machine to machine. The machine manufacturer may also provide different unbalance functions.



The centrifugal force that occurs basically depends on the rotational speed, the mass and the unbalance of the workpiece. A body with an uneven mass distribution that is put into rotary motion produces an unbalance. If the mass object is rotating, this creates outward-acting centrifugal forces. If the rotating mass is evenly distributed, the centrifugal forces cancel each other out. You compensate for the arising centrifugal forces by attaching balancing weights.

With Cycle **892 CHECK UNBALANCE** you define the maximum permissible unbalance and the maximum shaft speed. The control monitors these entries.

Further information: "Cycle 892 CHECK UNBALANCE ", Page 757

Unbalance monitor

The Unbalance Monitor function monitors the unbalance of a workpiece in turning mode. If a maximum unbalance limit specified by the machine manufacturer is exceeded, the control issues an error message and initiates an emergency stop.

In addition, you can further decrease the permissible unbalance limit by setting the optional machine parameter **limitUnbalanceUsr** (no. 120101). If this limit is exceeded, the control issues an error message. The control does not stop table rotation.

The control automatically activates the Unbalance Monitor function when you switch to turning mode. The unbalance monitor is active until you switch back to milling mode.

Further information: "Switching the operating mode with FUNCTION MODE", Page 234

Notes

⚠ WARNING

Caution: Danger to the operator and machine!

Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

- Clamp the workpiece in the spindle center
- Clamp workpiece securely
- Program low spindle speeds (increase as required)
- Limit the spindle speed (increase as required)
- Eliminate unbalance (calibrate)

- The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.
- The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps.
- To compensate an unbalance, several balancing weights at different positions may be required.

9.3 Grinding operations (option 156)

9.3.1 Fundamentals

Special types of milling machines allow performing both milling and grinding operations. A workpiece can thus be machined completely on one machine, even if complex milling and grinding operations are required.

Requirements

- Jig grinding (software option 156)
- Available kinematics description for jig grinding

  The machine manufacturer creates the kinematics description.

Production processes

The term grinding encompasses many types of machining that differ in quite a few respects, e.g.:

- Jig grinding
- Cylindrical grinding
- Surface grinding

The TNC7 currently features jig grinding.

Jig grinding is the grinding of a 2D contour. The tool movement in the plane is optionally superimposed by a reciprocation movement along the active tool axis.

Further information: "Jig grinding", Page 250

If grinding is enabled on your milling machine (option 156), the dressing function is also available. This means that you can shape or resharpen the grinding wheel in the machine.

Further information: "Dressing", Page 251

Reciprocating stroke

For jig grinding, the movement of the tool in the plane can be superimposed by a stroke movement, the so-called reciprocating stroke. The superimposed stroke movement is applied in the active tool axis.

You define an upper and a lower stroke limit and can start and stop the reciprocating stroke and reset the corresponding values. The reciprocating stroke is active until you stop it. **M2** or **M30** will stop the reciprocating stroke automatically.

The control provides cycles for defining, starting, and stopping reciprocating strokes.

As long as the reciprocating stroke is active in the program run, you cannot change to the other applications of the **Manual** operating mode.

The control presents the reciprocating stroke in the **Simulation** workspace of the **Program Run** operating mode.

Tools for grinding

When managing grinding tools, other geometric descriptions than those for milling or drilling tools are required. The control provides a special tool table for grinding and dressing tools. In tool management, the control displays only the required tool data for the current tool type.

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019

You can use compensation tables to change the values of grinding tools during program run.

Further information: "Tool compensation with compensation tables", Page 1120

Structure of an NC program for grinding

An NC program for grinding is structured as follows:

- Dressing of the grinding tool, if required
  Further information: "General information on the dressing cycles", Page 916
- Defining the reciprocating stroke
  Further information: "Cycle 1000 DEFINE RECIP. STROKE (option 156)", Page 911
- If necessary, explicitly starting the reciprocating stroke
  Further information: "Cycle 1001 START RECIP. STROKE (option 156)", Page 914
- Moving along the contour
- Stopping the reciprocating stroke
  Further information: "Cycle 1002 STOP RECIP. STROKE (option 156)", Page 915

You can use specific machining cycles (e.g., cycles for grinding, for machining pockets or studs, or SL cycles) to define the contour.

Further information: "Cycles for grinding", Page 909

9.3.2 Jig grinding

Application

On a milling machine, jig grinding will mainly be used for finishing a pre-machined contour with a grinding tool. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin or a grinding wheel. Jig grinding produces more precise results and a better surface quality than milling.

Related topics

- Cycles for grinding
  Further information: "Cycles for grinding", Page 909
- Tool data for grinding tools
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010
- Dressing of grinding tools
  Further information: "Dressing", Page 251

Requirements

- Jig grinding (software option 156)
- Available kinematics description for jig grinding
  The machine manufacturer creates the kinematics description.

Description of function

Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Grinding is also possible with a tilted working plane. The tool reciprocates along the active tool axis in the current working plane coordinate system (**WPL-CS**).

Notes

- The control does not support block scans while the reciprocating stroke is active.
  Further information: "Block scan for mid-program startup", Page 1965
- The reciprocating stroke continues to be active during a programmed **STOP** or **M0** as well as in **Single Block** mode even after the end of an NC block.
- If no cycle has been programmed and a contour is being ground whose smallest inside radius is smaller than the tool radius, the control will display an error message.
- If you machine with SL cycles, only those areas will be ground that are suitable for the given tool radius. In this case, the resulting contour will not be completely finished and may need to be reworked.

9.3.3 Dressing

Application

The term "dressing" refers to the sharpening or trueing up of a grinding tool inside the machine. During dressing, the dresser machines the grinding wheel. Thus, in dressing, the grinding tool is the workpiece.

Related topics

- Activating dressing mode with **FUNCTION DRESS**
  Further information: "Activating dressing mode with FUNCTION DRESS", Page 254
- Cycles for dressing
  Further information: "General information on the dressing cycles", Page 916
- Tool data for dressing tools
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019
- Jig grinding
  Further information: "Jig grinding", Page 250

Requirements

- Jig grinding (software option 156)
- Available kinematics description for jig grinding
  The machine manufacturer creates the kinematics description.

Description of function

In dressing, the workpiece datum is located on an edge of the grinding wheel. Select the respective edge by using Cycle **1030 ACTIVATE WHEEL EDGE**.

During dressing, the axes are arranged such that the X coordinates describe positions on the radius of the grinding wheel, and the Z coordinates describe the positions along the axis of the grinding wheel. The dressing programs are thus not contingent on the machine type.

The machine manufacturer defines which machine axes will perform the programmed movements.

The dressing operation removes material from the grinding wheel and may cause wear of the dressing tool. The material removal and wear lead to changed tool data that need to be compensated for after dressing.

The **COR_TYPE** parameter provides the following compensation options for the tool data:

- **Grinding wheel with compensation, COR_TYPE_GRINDTOOL**

  Compensation method removing material from the grinding tool

  Further information: "Stock removal on the grinding tool", Page 253
- **Dressing tool with wear, COR_TYPE_DRESSTOOL**

  Compensation method removing material from the dresser

  Further information: "Stock removal on the grinding tool", Page 253

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

Use the Cycles **1032 GRINDING WHL LENGTH COMPENSATION** and **1033 GRINDING WHL RADIUS COMPENSATION** to compensate the grinding wheel or the dresser, regardless of the compensation method.

Further information: "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962

Further information: "Cycle 1033 GRINDING WHL RADIUS COMPENSATION (option 156)", Page 964

Simplified dressing with a macro

Your machine manufacturer can program the entire dressing mode in a macro.

In this case, the machine manufacturer determines the dressing sequence. It is not necessary to program **FUNCTION DRESS BEGIN**.

Depending on this macro, you can start the dressing mode with one of the following cycles:

- Cycle **1010 DRESSING DIAMETER**
- Cycle **1015 PROFILE DRESSING**
- Cycle **1016 DRESSING OF CUP WHEEL**
- OEM cycle

Compensation methods

Stock removal on the grinding tool

During dressing, a dressing tool is usually used that is harder than the grinding tool. Due to the difference in hardness, the stock removal during dressing mainly takes place at the grinding tool. The programmed dressing amount is actually removed at the grinding tool, since the dressing tool does not noticeably wear.
In this case the compensation method **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** is used in the **COR_TYPE** parameter of the grinding tool.

Further information: "Tool management ", Page 297

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

With this compensation method, the tool data of the dressing tool remain constant. The control compensates only for the grinding tool:

- Programmed dressing amount in the basic data of the grinding tool, e.g. **R-OVR**
- If applicable, measured deviation between nominal and actual dimension in the compensation data of the grinding tool, e.g. **dR-OVR**

Stock removal on dressing tool

In contrast to the standard situation, stock removal does not take place only on the grinding tool in certain grinding and dressing combinations. In this case the dressing tool wears noticeably, e.g. with very hard grinding tools in combination with softer dressing tools. To compensate for this noticeable wear on the dressing tool, the control offers the compensation method **Dressing tool with wear, COR_TYPE_DRESSTOOL** in the **COR_TYPE** parameter of the dressing tool.

Further information: "Tool management ", Page 297

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

With this compensation method the tool data of the dressing tool change significantly. The control compensates for both the grinding tool and the dressing tool:

- Dressing amount in the basic data of the grinding tool, e.g. **R-OVR**
- Measured wear in the compensation data of the dressing tool, e.g. **DXL**

If you use the compensation method **Dressing tool with wear, COR_TYPE_DRESSTOOL**, the control stores the tool number of the dressing tool used in the **T_DRESS** parameter of the grinding tool after dressing. During future dressing processes, the control monitors whether the defined dressing tool is used. If you use a different dressing tool, the control interrupts the dressing with an error message.

You must recalibrate the grinding tool after each dressing process so that the control can determine and compensate for the wear.

Notes

- For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.
- Measure the grinding tool after dressing so that the control enters the correct delta values.
- Not all grinding tools require dressing. Comply with the information provided by your tool manufacturer.
- When using the **Dressing tool with wear, COR_TYPE_DRESSTOOL** correction method, inclined dressing tools must not be used.

9.3.4 Activating dressing mode with FUNCTION DRESS

Application

With **FUNCTION DRESS** you activate a dressing kinematic model for dressing a grinding tool. The grinding tool is then the workpiece and the axes may move in the opposite direction.

Your machine manufacturer might provide a simplified dressing procedure.

Further information: "Simplified dressing with a macro", Page 252

Related topics

- Cycles for dressing

  Further information: "General information on the dressing cycles", Page 916
- Fundamentals of dressing

  Further information: "Dressing", Page 251

Requirements

- Jig grinding (software option 156)
- Available kinematics description for dressing

  The machine manufacturer creates the kinematics description.
- Grinding tool is inserted
- Grinding tool without assigned tool-carrier kinematics

Description of function

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

For the control to switch to the kinematic model for dressing, you must program the dressing process between the functions **FUNCTION DRESS BEGIN** and **FUNCTION DRESS END**.

If dressing mode is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 163

You can switch back to normal operation with the function **FUNCTION DRESS END**.

In the event of an NC program abort or a power interruption, the control automatically activates normal operation and the kinematic model that was active prior to dressing mode.

Input

11 FUNCTION DRESS BEGIN "Dress"	; Activate dressing mode with the **Dress** kinematics

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION DRESS	Syntax initiator for dressing mode
BEGIN or **END**	Activate or deactivate dressing mode
Name or **QS**	Name of the selected kinematic model Fixed or variable name Only if **BEGIN** has been selected Optional syntax element

Notes

NOTICE
Danger of collision!
The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!
▶ Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
▶ Make sure there is no risk of collision
▶ Verify the NC program by slowly executing it block by block

NOTICE
Danger of collision!
With an active kinematic model, the machine movements may be in the opposite direction. There is a risk of collision when moving the axes!
▶ In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
▶ If necessary, program a kinematic switch-over

- During dressing, the cutting edge of the dresser must be at the same height as the grinding wheel. The programmed Y coordinate must be 0.
- With the switch to dressing mode, the grinding tool remains in the spindle and retains its current rotational speed.
- The control does not support a block scan during the dressing process. If, during a block scan, you select the first NC block after the dressing operation, then the control moves to the most recently approached position in the dressing operation.

  Further information: "Block scan for mid-program startup", Page 1965
- If the “tilt working plane” function or **TCPM** function is active, then you cannot switch to dressing mode.
- The control resets the manual tilting functions (option 8) and **FUNCTION TCPM** (option 9) when dressing mode is activated.

  Further information: "3-D rotation window (option 8)", Page 1098
  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
- In dressing mode you can use **TRANS DATUM** to change the workpiece datum. No other NC functions or coordinate conversion cycles are permitted in dressing mode. The control displays an error message.

  Further information: "Datum shift with TRANS DATUM", Page 1046
- The **M140** function is not allowed in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation. The times determined by the simulation do not reflect the actual machining times. One reason for this is the necessary switching of the kinematic model.

10

Workpiece Blank

10.1 Defining a workpiece blank with BLK FORM

Application

You use the **BLK FORM** function to define a workpiece blank for graphic simulation of the NC program.

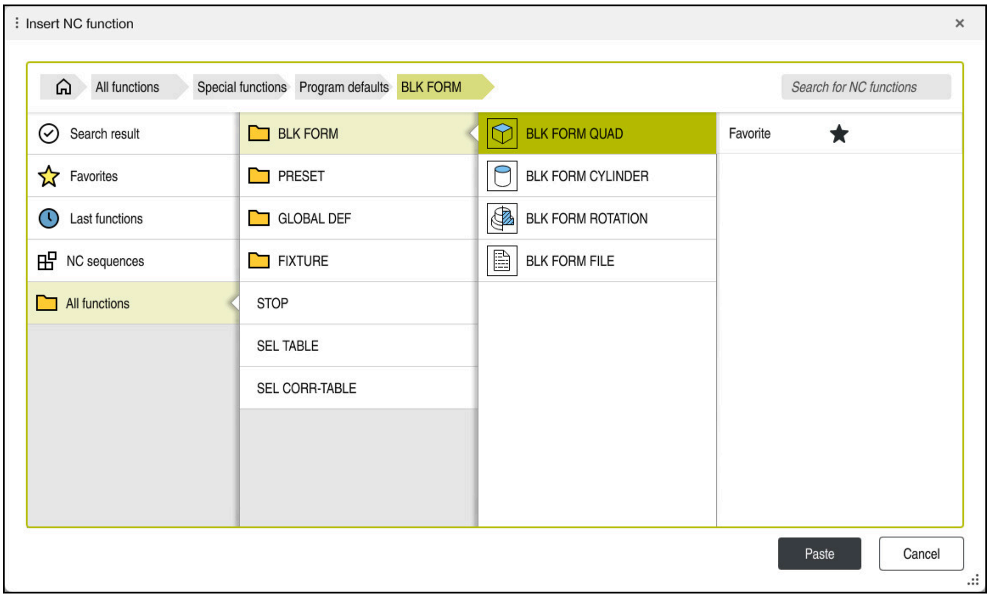
Related topics

- Depiction of the workpiece blank in the **Simulation** workspace
  Further information: "Simulation Workspace", Page 1535
- Blank form update with **FUNCTION TURNDATA BLANK** (option 50)
  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1124

Description of function

You define the blank relative to the workpiece preset.

Further information: "Presets in the machine", Page 210



The **Insert NC function** window for workpiece blank definition

When you create a new NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

Further information: "Creating a new NC program", Page 132

The control offers the following workpiece blank definitions:

Icon	Function	Further information
	**BLK FORM QUAD** Cuboid workpiece blank	Page 261
	**BLK FORM CYLINDER** Cylindrical workpiece blank	Page 262
	**BLK FORM ROTATION** Rotationally symmetric blank with a defin-able contour	Page 263
	BLK FORM FILE STL file as workpiece blank and finished part	Page 264

Notes

NOTICE

Danger of collision!

Even if Dynamic Collision Monitoring (DCM) is active, the control does not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

- Enable the **Advanced checks** toggle switch for simulations
- Check the machining sequence using a simulation
- Carefully test your NC program or program section in the **Single Block** mode

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.

- There are various ways for selecting files or subprograms:
  - Enter the file path
  - Enter the number or name of the subprogram
  - Select the file or subprogram by means of a selection window
  - Define the file path or name of the subprogram in a QS parameter
  - Define the number of the subprogram in a Q, QL or QR parameter

  If the called file is located in the same directory as the calling NC program, it might be sufficient to enter just the file name.
- 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 control displays the workpiece blank in the simulation only after having processed the entire workpiece blank definition.
- Even if you have closed the **Insert NC function** window or want to add a workpiece blank definition after writing an NC program, you can always define a workpiece blank via the **Insert NC function** window.
- 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: "Advanced checks in the simulation", Page 1190
- In the **Simulation** workspace you can export the current view of the workpiece as an STL file. This function allows you to create missing 3D models, for example semifinished parts if there are several machining steps.

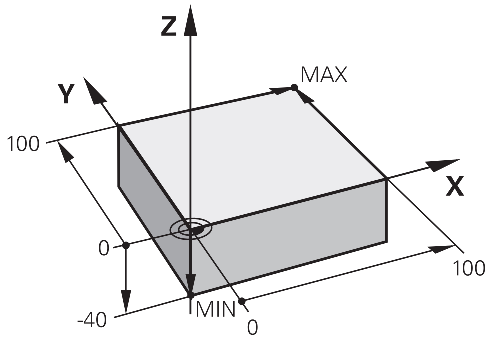
  Further information: "Exporting a simulated workpiece as STL file", Page 1546

10.1.1 Cuboid workpiece blank with BLK FORM QUAD

Application

With **BLK FORM QUAD** you define a cuboid workpiece blank. You use a MIN point and a MAX point to define a spatial diagonal.

Description of function



Cuboid workpiece blank with MIN point and MAX point

The sides of the cuboid are parallel to the **X**, **Y** and **Z** axes.

You define the cuboid by entering a MIN point for the bottom front left corner and a MAX point for the top rear right corner.

You define the coordinates of the points in the **X**, **Y** and **Z** relative to the workpiece preset. If you define a positive value for the MAX point in the Z coordinate, the blank is given an oversize.

Further information: "Presets in the machine", Page 210

If you use a cuboid workpiece blank for turning (option 50), keep the following in mind:

Even if the turning operation takes place in a two-dimensional plane (Z and X coordinates), you have to program the Y values for a rectangular blank in the definition of the workpiece blank.

Further information: "Fundamentals", Page 236

Input

1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Cuboid workpiece blank

The NC function includes the following syntax elements:

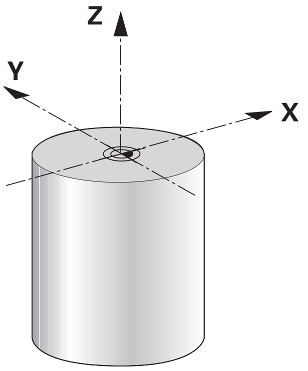
Syntax element	Meaning
BLK FORM	Start of syntax for cuboid workpiece blank
0.1	Designation of the first NC block
Z	Tool axis Other possibilities might be available, depending on the machine.
X Y Z	Coordinate definition of the MIN point
0.2	Designation of the second NC block
X Y Z	Coordinate definition of the MAX point

10.1.2 Cylindrical workpiece blank with BLK FORM CYLINDER

Application

With **BLK FORM CYLINDER** you define a cylindrical workpiece blank. You can define a cylinder either as a solid piece or as a hollow pipe.

Description of function



Cylindrical blank

To define the cylinder, enter at least the radius or diameter and the height.

The workpiece preset is in the cylinder center in the working plane. Optionally you can define an oversize and the inside radius or diameter of the blank.

Input

1 BLK FORM CYLINDER Z R50 L105 DIST +5 RI10	; Cylindrical blank

The NC function includes the following syntax elements:

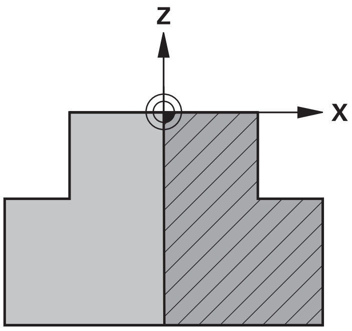
Syntax element	Meaning
BLK FORM CYLINDER	Syntax initiator for cylindrical workpiece blank
Z	Tool axis Other possibilities might be available, depending on the machine.
R or **D**	Radius or diameter of the cylinder
L	Total height of the cylinder
DIST	Oversize of the cylinder relative to the workpiece preset Optional syntax element
RI or **DI**	Inside radius diameter of the core hole Optional syntax element

10.1.3 Rotationally symmetric workpiece blank with BLK FORM ROTATION

Application

With **BLK FORM ROTATION** you define a rotationally symmetric workpiece blank with a definable contour. You define the contour in a subprogram or separate NC program.

Description of function



Blank contour with tool axis **Z** and main axis **X**

In the workpiece blank definition you refer to the contour description.

In the contour description, you program a half-section of the contour around the tool axis as the rotational axis.

The following conditions apply to the contour description:

- Only coordinates of the main axis and tool axis
- Starting point defined in both axes
- Closed contour
- Only positive values in the main axis
- Positive and negative values are possible in the tool axis

The workpiece preset is in the center of the blank in the working plane. You define the coordinates of the blank contour relative to the workpiece preset. You can also define an oversize.

Input

1 BLK FORM ROTATION Z DIM_R LBL "BLANK"	; Rotationally symmetric blank
*** - ...**	
11 LBL "BLANK"	; Subprogram start
12 L X+0 Z+0	; Beginning of contour
13 L X+50	; Coordinates in positive direction of main axis
14 L Z+50	
15 L X+30	
16 L Z+70	
17 L X+0	
18 L Z+0	; End of contour
19 LBL 0	; End of subprogram

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM ROTATION	Syntax initiator for rotationally symmetric workpiece blank
Z	Active tool axis Other possibilities might be available, depending on the machine.
DIM_R or **DIM_D**	Interpret values in the main axes in the contour description as radius or diameter
LBL or **FILE**	Name or number of the contour subprogram or path of the separate NC program

Notes

- If you program the contour description with incremental values, the control interprets the values as radii regardless of whether **DIM_R** or **DIM_D** is selected.
- With CAD Import (software option 42), you can load contours from CAD files and save them in subprograms or separate NC programs.
  Further information: "Opening CAD Files with the CAD-Viewer", Page 1455

10.1.4 STL file as workpiece blank with BLK FORM FILE

Application

You can integrate 3D models in STL format as workpiece blank and optionally as finished part. This function is particularly convenient in combination with CAM programs, where the required 3D models are available in addition to the NC program.

Requirement

- Max. 20 000 triangles per STL file in ASCII format
- Max. 50 000 triangles per STL file in binary format

Description of function

The dimensions of the NC program come from the same source as the dimensions of the 3D model.

Input

1 BLK FORM FILE "TNC:\CAD\blank.stl" TARGET "TNC:\CAD\finish.stl"	; STL file as workpiece blank and finished part

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM FILE	Syntax initiator for an STL file as workpiece blank
" "	Path of the STL file
TARGET	STL file as finished part Optional syntax element
" "	Path of the STL file

Notes

- In the **Simulation** workspace you can export the current view of the workpiece as an STL file. This function allows you to create missing 3D models, for example semifinished parts if there are several machining steps.

  Further information: "Exporting a simulated workpiece as STL file", Page 1546
- If you have integrated a workpiece blank and a finished part, you can compare the models in the simulation and easily identify any residual material.

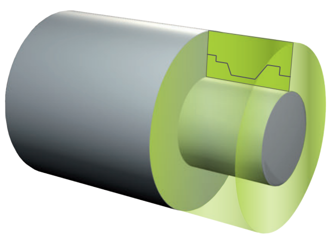
  Further information: "Model comparison", Page 1551
- The control loads STL files in binary format faster than STL files in ASCII format.

10.2 Blank form update in turning mode with FUNCTION TURNDATA BLANK (option 50)

Application

Using the blank form update feature, the control detects the already machined areas and adapts all approach and departure paths to the specific, current machining situation. Thus, air cuts are avoided and the machining time is significantly reduced.

You define the workpiece blank for blank form update in a subprogram or separate NC program.



Related topics

- Subprograms

  Further information: "Subprograms and program section repeats with the label LBL", Page 384
- Turning mode: **FUNCTION MODE TURN**

  Further information: "Fundamentals", Page 236
- Defining a workpiece blank with **BLK FORM** for simulation

  Further information: "Defining a workpiece blank with BLK FORM", Page 258

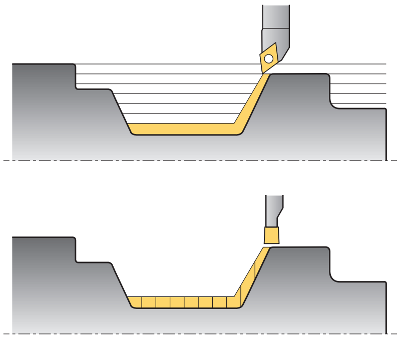
Requirements

- Combined milling/turning (software option 50)
- **FUNCTION MODE TURN** must be active

  Blank form update is only possible with cycle machining in turning mode.
- Closed blank contour for blank form updating

  The starting and end positions must be identical. The workpiece blank corresponds to the cross-section of a rotationally symmetrical body.

Description of function



With **TURNDATA BLANK** you call a contour description used by the control as an updated workpiece blank.

You can define the workpiece blank in a subprogram within the NC program or as a separate NC program.

Blank form update is only active in conjunction with roughing cycles. In finishing cycles the control always machines the entire contour, for example so that the contour does not have any offset.

Further information: "Cycles for milling and turning", Page 744

There are various ways for selecting files or subprograms:

- Enter the file path
- Enter the number or name of the subprogram
- Select the file or subprogram by means of a selection window
- Define the file path or name of the subprogram in a QS parameter
- Define the number of the subprogram in a Q, QL or QR parameter

Use **FUNCTION TURNDATA BLANK OFF** to deactivate blank form update.

Input

1 FUNCTION TURNDATA BLANK LBL "BLANK"	; Blank form update with a workpiece blank from the subprogram "BLANK"
*** - ...**	
11 LBL "BLANK"	; Subprogram start
12 L X+0 Z+0	; Beginning of contour
13 L X+50	; Coordinates in positive direction of main axis
14 L Z+50	
15 L X+30	
16 L Z+70	
17 L X+0	
18 L Z+0	; End of contour
19 LBL 0	; End of subprogram

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA BLANK	Syntax initiator for blank form update in turning mode
OFF, **File**, **QS** or **LBL**	Deactivate blank form update, blank contour as separate NC program, or call as subprogram
Number, **Name** or **QS**	Number or name of the separate NC programor subprogram Fixed or variable number or name If **File**, **QS** or **LBL** is selected

11

Tools

11.1 Fundamentals

To use the control's functions, you must define the tools for the control using real data (e.g., the radius). This makes programming easier and improves process reliability.

To add a tool to the machine, follow the sequence below:

- Prepare your tool and clamp the tool into a suitable tool holder.
- To measure the tool dimensions, starting from the tool carrier preset, measure the tool (e.g., using a tool presetter). The control needs these dimensions for calculating the paths.

  Further information: "Tool carrier reference point", Page 271
- Further tool data are needed to completely define the tool. Take these tool data from the manufacturer’s tool catalog, for example.

  Further information: "Tool data for the tool types", Page 284
- Save all collected tool data of this tool in the tool management.

  Further information: "Tool management ", Page 297
- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

  Further information: "Tool carrier management", Page 301
- After finishing tool definition, program a tool call within an NC program.

  Further information: "Tool call by TOOL CALL", Page 304
- If your machine is equipped with a chaotic tool changer system and a double gripper, the tool change time may be shortened by pre-selecting the tool.

  Further information: "Tool pre-selection by TOOL DEF", Page 311
- If needed, perform a tool usage test before starting the program. This process checks if the tools are available in the machine and have sufficient remaining tool life.

  Further information: "Tool usage test", Page 312
- After machining a workpiece and measuring it, you may correct the tools.

  Further information: "Tool radius compensation", Page 1114

11.2 Presets on the tool

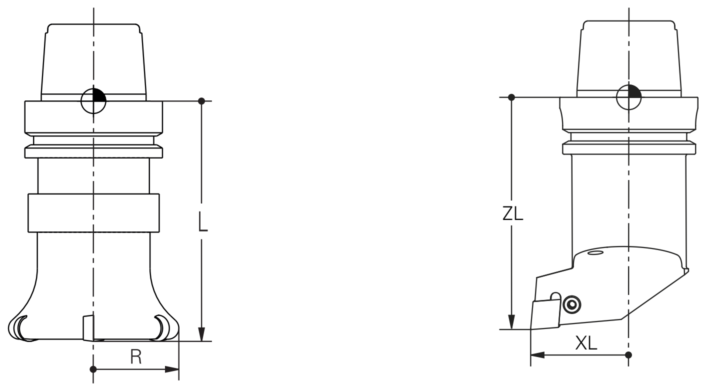
The control distinguishes the following presets on the tool for different calculations or applications.

Related topics

- Presets in the machine or on the workpiece

  Further information: "Presets in the machine", Page 210

11.2.1 Tool carrier reference point

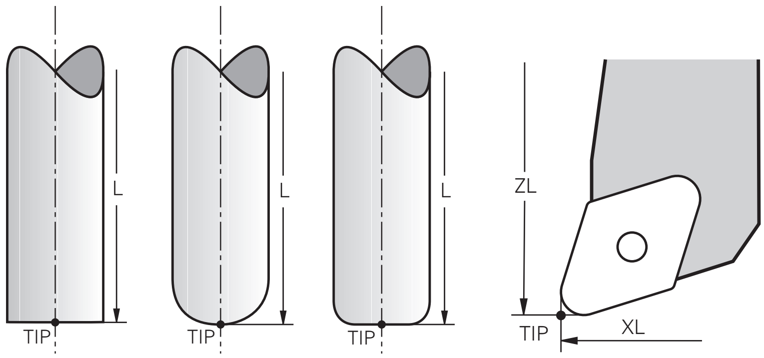


The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

Starting from the tool carrier reference point, define the tool dimensions in the tool management (e.g., length **L** and radius **R**).

Further information: "Tool management ", Page 297

11.2.2 Tool tip TIP



The tool tip has the greatest distance from the tool carrier reference point. The tool tip is the origin of the tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 1022

In case of milling cutters, the tool tip is at the center of the tool radius **R** and at the longest point of the tool on the tool axis.

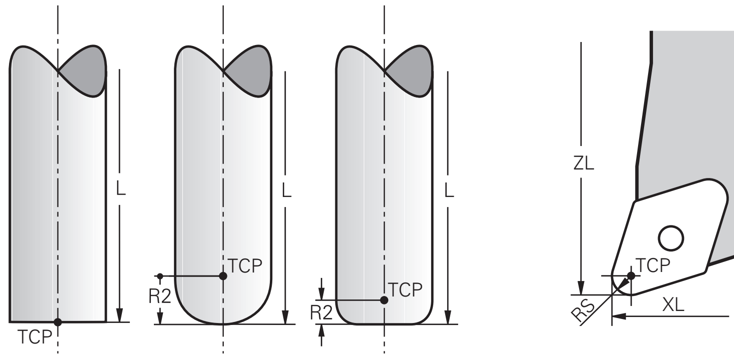
You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

- **L**
- **DL**
- **ZL** (option 50, option 156)
- **XL** (option 50, option 156)
- **YL** (option 50, option 156)
- **DZL** (option 50, option 156)
- **DXL** (option 50, option 156)
- **DYL** (option 50, option 156)
- **LO** (option 156)
- **DLO** (option 156)

Further information: "Tool data for the tool types", Page 284

In the case of lathe tools (option 50) the control uses the theoretical tool tip, i. e. the longest measured values **ZL**, **XL** and **YL**.

11.2.3 Tool center point (TCP, tool center point)



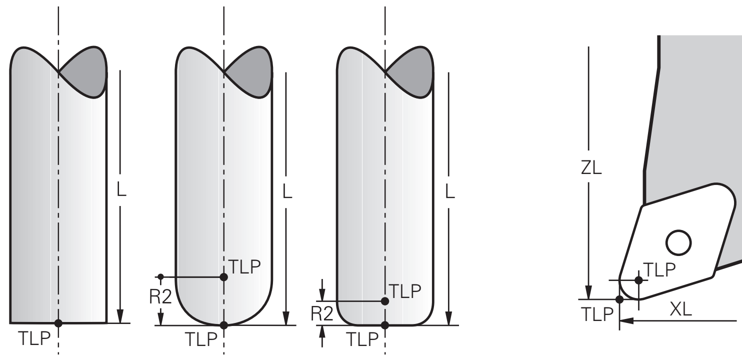
The tool center point is the center of the tool radius **R**. If a second tool radius (**R2**) is defined, the tool center point is offset from the tool tip by this value.

In the case of turning tools (option 50), the tool center point is at the center of cutter radius **RS**.

Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: "Tool data for the tool types", Page 284

11.2.4 Tool location point (TLP, tool location point)

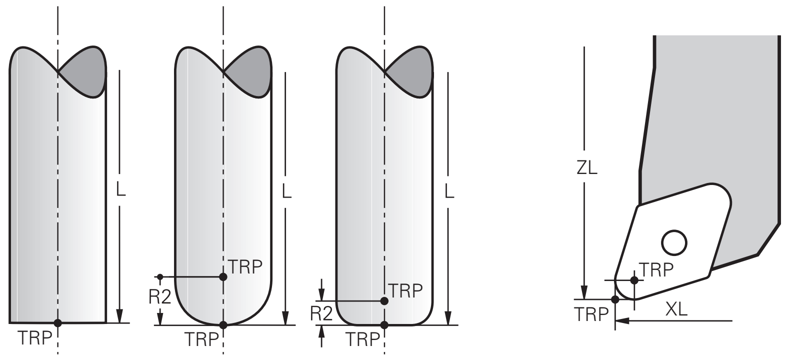


The control positions the tool on the tool location point. By default, the tool location point is at the tool tip.

In the **FUNCTION TCPM** function (option 9), you can also choose the tool location point to be at the tool center point.

Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

11.2.5 Tool rotation point (TRP, tool rotation point)



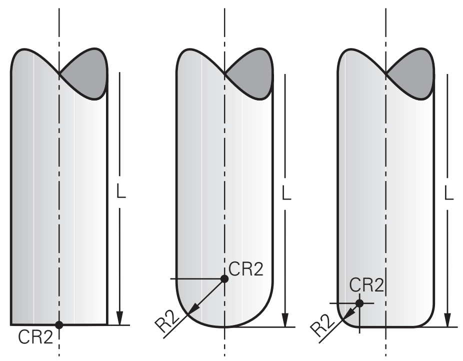
When applying the tilting function with **MOVE** (option 8), the control tilts around the tool rotation point. By default, the tool rotation point is at the tool tip.

When selecting **MOVE** in **PLANE** functions, the syntax element **DIST** is used to define the relative position between the workpiece and the tool. The control shifts the tool rotation point from the tool tip by this value. When **DIST** is not defined, the control keeps the tool tip constant.

Further information: "Rotary axis positioning", Page 1088

In the **FUNCTION TCPM** function (option 9), you can also choose the tool rotation point to be at the tool center point.

11.2.6 Tool radius 2 center (CR2, center R2)



The control uses the tool radius 2 center in conjunction with 3D tool compensation (option 9). In the case of straight lines **LN**, the surface normal vector points to that point and defines the direction of the 3D tool compensation.

Further information: "3D tool compensation (option 9)", Page 1126

The tool radius 2 center is offset from the tool tip and the cutting edge by the **R2** value.

11.3 Tool data

11.3.1 Tool ID number

Application

Each tool has a unique number which equals the row number of the tool management. Each tool ID number is unique.

Further information: "Tool management ", Page 297

Description of function

The tool ID numbers can be defined in a range from 0 to 32,767.

The tool with the number 0 is defined as the zero tool with the length and the radius 0. Upon a TOOL CALL 0, the control unloads the currently used tool and inserts no new tool.

Further information: "Tool call", Page 304

11.3.2 Tool name

Application

A tool name can be assigned in addition to the tool ID number. Contrary to the tool ID number, a tool name is not unique.

Description of function

The tool name allows identifying tools easier within the tool management. To this end, key features can be defined such as the diameter or the type of machining (e.g., **MILL_D10_ROUGH**).

As tool names are not unique, assign names that clearly identify the tools.

A tool name may contain up to 32 characters.

Permitted characters

You can use the following characters for the tool name:

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 # $ % & , - _ .

When entering lowercase letters, the control will substitute them by uppercase letters upon saving.

Note

- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:

  - 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 least remaining tool life.

11.3.3 Database ID

Application

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 database ID is entered in the **DB_ID** column of the tool management.

Related topics

- **DB_ID** column of tool management

  Further information: "Tool table tool.t", Page 1995

Description of function

The database ID is stored in the **DB_ID** column of the tool management.

For indexed tools, you can define the database ID either only for the physically existing main tool or as an ID for the data record at each index.

For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool.

Further information: "Indexed tool", Page 276

A database ID may contain a maximum of 40 characters and is unique in the tool management.

The control does not allow a tool call with the database ID.

11.3.4 Indexed tool

Application

Using an indexed tool, several different sets of tool data can be stored for one physically available tool. This feature enables indication of a certain point on the tool by means of the NC program which does not necessarily have to correspond with the maximum tool length.

Description of function

Tools with multiple lengths and radii cannot be defined in one row of the tool management table. Additional table rows are required, specifying the full definitions of the indexed tools. The lengths of the indexed tools, starting from the maximum tool length, approach the tool carrier preset as the index increases.

Further information: "Tool carrier reference point", Page 271

Further information: "Creating an indexed tool", Page 277

Examples of an application of indexed tools:

- Step drill

  The tool data of the main tool contain the drill tip, which corresponds to the maximum length. The tool steps are defined are indexed tools. This makes the lengths equal the actual tool dimensions.
- NC center drill

  The main tool is used for defining the theoretical tool tip as the maximum length. This can be used for centering, for example. The indexed tool defines a point along the tool tooth. This can be used for deburring, for example.
- Cut-off milling cutter or T-slot milling cutter

  The main tool is used for defining the lower point of the cutting edge, which equals the maximum length. The indexed tool defines the upper point of the cutting edge. When using the indexed tool for cutting-off, the specified workpiece height can be directly programmed.

Creating an indexed tool

To create an indexed tool:

- Select the **Tables** operating mode
- Select **Tool management**
- Activate **Edit**
- > The control enables tool management for editing.
- Select **Insert tool**
- > The control opens the **Insert tool** pop-up window.
- Define the tool type
- Define the tool number of the main tool (e.g., **T5**)
- Press **OK**
- > The control adds table row **5**.
- Define all necessary tool data including the maximum tool length.

  Further information: "Tool data for the tool types", Page 284
- Select **Insert tool**
- > The control opens the **Insert tool** pop-up window.
- Define the tool type
- Define the tool number of the indexed tool (e.g., **T5.1**)

> The main tool number and an index after the dot define an indexed tool.

- Press **OK**
- > The control adds table row **5.1**.
- Define all required tool data

  Further information: "Tool data for the tool types", Page 284

> The control does not adopt any main tool data!
>
> The lengths of the indexed tools approach the tool carrier preset as the index rises, starting from the maximum tool length.
>
> **Further information:** "Tool carrier reference point", Page 271

Notes

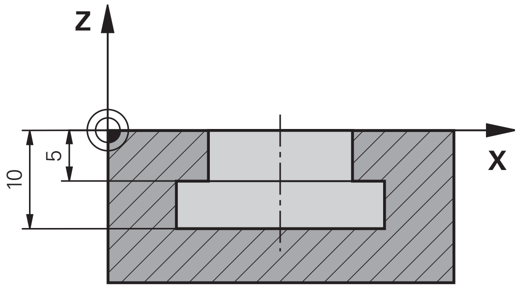
- The control describes some parameters automatically (e.g., the current tool age **CUR_TIME**). The control describes these parameters separately for each table row.

  Further information: "Tool table tool.t", Page 1995
- Index numbers do not need to be sequential. It is possible, for example, to create the tools **T5**, **T5.1** and **T5.3**.
- Up to nine indexed tools can be added to each main tool.

  When defining a replacement tool **RT**, this applies to the respective table row exclusively. When an indexed tool is worn and consequently blocked, this also does not apply to all other indices. This ensures, for example, that the main tool can still be used.

  Further information: "Automatically inserting a replacement tool with M101", Page 1355

Example of T-slot milling cutter



In this example, you program a T-slot with dimensions referring to the top and bottom edges as viewed from the coordinates surface. The height of the T-slot is larger than the length of the cutting edge of the tool used. This requires two steps.

Two tool definitions are required for producing the T-slot.

- The main tool dimension refers to the lower point of the cutting edge, which equals the maximum tool length. This can be used for machining the bottom edge of the T-slot.
- The dimension of the indexed tool refers to the upper point of the cutting edge. This can be used for machining the top edge of the T-slot.

> Please ensure that all required tool data are defined both for the main tool and for the indexed tool! In case of a rectangular tool, the radius remains identical in both table lines.

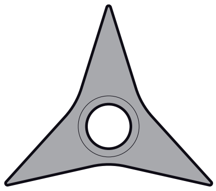
The T-slot is programmed in two machining steps:

- The 10 mm depth is programmed with the main tool.
- The 5 mm depth is programmed with the indexed tool.

11 TOOL CALL 7 Z S2000	; Call the main tool
12 L X+0 Y+0 Z+10 R0 FMAX	; Pre-position the tool
13 L Z-10 R0 F500	; Move to machining depth
14 CALL LBL "CONTOUR"	; Machine the bottom edge of the T-slot with the main tool
*** - ...**	
21 TOOL CALL 7.1 Z F2000	; Call the indexed tool
22 L X+0 Y+0 Z+10 R0 FMAX	; Pre-position the tool
23 L Z-5 R0 F500	; Move to machining depth
24 CALL LBL "CONTOUR"	; Machine the top edge of the T-slot with the indexed tool

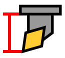
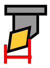
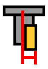
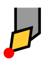
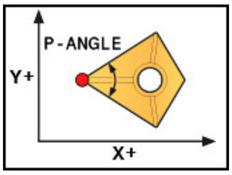
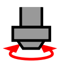
Example of a FreeTurn tool

You need the following tool data for a FreeTurn tool:



FreeTurn tool with three finishing teeth

> Integrating information about the point angles **P-ANGLE** and the tool length **ZL**, e.g. **FT1_35-35-35_100**, into the tool name is recommended.

Icon and parameter	Meaning	Intended use
 **ZL**	Tool length 1	The tool length **ZL** equals the total tool length, relating to the tool carrier preset. **Further information:** "Presets on the tool", Page 271
 **XL**	Tool length 2	The tool length **XL** equals the difference between the spindle center and the tool tip of the tooth. **XL** must always be defined as a negative value with FreeTurn tools. **Further information:** "Presets on the tool", Page 271
 **YL**	Tool length 3	The tool length **YL** is always 0 with FreeTurn tools.
 **RS**	Cutting radius	You can take the radius **RS** from the tool catalog.
 **TYPE**	Lathe tool type	You select between a rough-turning tool (**ROUGH**) and finishing tool (**FINISH**). **Further information:** "Subgroups of technology-specific tool types", Page 282
 **TO**	Tool orientation	The tool orientation **TO** is always 18 with FreeTurn tools. 
 **ORI**	Angle of orientation	The angle of orientation **ORI** defines the offset of the single teeth with respect to one another. If the first tooth has the value 0, define the second tooth of symmetrical tools at 120 and the third tooth at 240.

Icon and parameter	Meaning	Intended use
 **P-ANGLE**	Point angle	You can get the point angle **P-ANGLE** from the tool catalog.
 **CUTLENGTH**	Cutting-edge length	You can get the tooth length **CUTLENGTH** from the tool catalog.
	Toolcarrier kinematics	Using the optional tool-carrier kinematics, the control can monitor the tool for collisions, for example. Assign the same kinematics to each single tooth.

11.3.5 Tool types

Application

Depending on the selected tool type, the control displays the editable tool data in the tool management.

Related topics

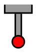
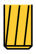
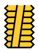
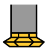
- Editing the tool data in the tool management
  Further information: "Tool management ", Page 297

Description of function

A number is additionally assigned to each tool type.

The following tool types can be selected in the **TYPE** column of the tool management:

Icon	Tool type	Number
	Milling cutter (**MILL**)	0
	Rough cutter (**MILL_R**)	9
	Finishing cutter (**MILL_F**)	10
	Face mill (**MILL_FACE**)	14
	Ball-nose cutter (**BALL**)	22
	Toroid cutter (**TORUS**)	23
	Chamfer mill (**MILL_CHAMFER**)	24
	Drill (**DRILL**)	1
	Tap (**TAP**)	2
	NC center drill (**CENT**)	4

Icon	Tool type	Number
	Turning tool (**TURN**) **Further information:** "Types within the turning tools", Page 283	29
	Touch probe (**TCHP**)	21
	Reamer (**REAM**)	3
	Countersink (**CSINK**)	5
	Piloted counterbore (**TSINK**)	6
	Boring tool (**BOR**)	7
	Back boring tool (**BCKBOR**)	8
	Thread miller (**GF**)	1
	Thread miller with chamfer (**GSF**)	16
	Thread mill with single thread (**EP**)	17
	Thread mill with indexable insert (**WSP**)	18
	Thread drilling/milling cutter (**BGF**)	19
	Circular thread mill (**ZBGF**)	20
	Grinding wheel (**GRIND**) **Further information:** "Types within the grinding tools", Page 283	30
	Dressing tool (**DRESS**) **Further information:** "Types within the dressing tools", Page 283	31

These tool types allow filtering the tools in the tool management.

Further information: "Tool management ", Page 297

Subgroups of technology-specific tool types

In the **TYPE** column of the tool management, a technology-specific tool type can be defined, depending on the selected tool type. The control offers the **TYPE** column for the **TURN**, **GRIND** and **DRESS** tool types. Specify the tool type more precisely within these technologies.

Types within the turning tools

Select between the types below within the turning tools:

Icon	Tool type	Number
	Rough-turning tool (**ROUGH**)	11
	Finish-turning tool (**FINISH**)	12
	Thread-turning tool (**THREAD**)	14
	Recessing tool (**RECESS**)	15
	Button tool (**BUTTON**)	21
	Recess-turning tool (**RECTURN**)	26

Types within the grinding tools

Select between the types below within the grinding tools:

Icon	Tool type	Number
	Cylindrical grinding pin (**GRIND_PIN**)	1
	Conical grinding pin (**GRIND_CONE**)	2
	Cup wheel (**GRIND_CUP**)	3
	Straight wheel (**GRIND_CYLINDER**) Currently no function	26
	Slant wheel (**GRIND_ANGULAR**) Currently no function	27
	Facing wheel (**GRIND_FACE**) Currently no function	28

Types within the dressing tools

Select between the types below within the dressing tools:

Icon	Tool type	Number
	Stationary dresser with radius (**DRESS_FIX_RADIUS**)	101
	Horn-type dresser (**HORNED**) Currently no function	102
	Rotating dresser with radius (**DRESS_ROT_RADIUS**)	103
	Stationary dresser (flat) (**DRESS_FIX_FLAT**)	110
	Rotating (flat) (**DRESS_ROT_FLAT**)	120

11.3.6 Tool data for the tool types

Application

The tool data provide the control with all information necessary for calculating and checking the required movements.

The necessary data depend on the technology and the tool type.

Related topics

- Editing the tool data in the tool management

  Further information: "Tool management ", Page 297
- Tool types

  Further information: "Tool types", Page 281

Description of function

Some of the necessary tool data can be determined using the following options:

- You can measure your tools in the machine (e. g., with a tool touch probe) or externally with a tool presetter.

  Further information: "Touch Probe Cycles: Automatic Tool Measurement", Page 1906
- Take further tool information from the manufacturer's tool catalog (e.g., the material or the number of teeth).

In the tables below, the relevance of the parameters is sub-divided into the optional, recommended and required categories.

The control takes recommended parameters into account for at least one of the functions below:

- Simulation

  Further information: "Simulation of tools", Page 1544
- Machining or touch probe cycles

  Further information: "Machining Cycles", Page 475

  Further information: "Programmable Touch Probe Cycles", Page 1589
- Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164

Tool data for milling and drilling tools

The control offers the following parameters for milling and drilling tools:

Icon and parameter	Meaning	Intended use
L	Length	Required for all milling and drilling tool types
R	Radius	Required for all milling and drilling tool types
R2	Radius 2	Required for the following milling and drilling tool types: ■ **Ball-nose cutter** ■ **Toroid cutter**
DL	Delta value of length	Optional The control describes this parameter in connection with touch probe cycles.
DR	Delta value of radius	Optional The control describes this parameter in connection with touch probe cycles.
DR2	Delta value of radius 2	Optional The control describes this parameter in connection with touch probe cycles.
LCUTS	Tooth length	Recommended
RCUTS	Tooth width	Recommended
LU	Useful length	Recommended
RN	Neck radius	Recommended
ANGLE	Plunge angle	Recommended for the following milling and drilling tool types: ■ **Milling tool** ■ **Roughing mill** ■ **Finishing cutter** ■ **Ball-nose cutter** ■ **Toroid cutter**

Icon and parameter	Meaning	Intended use
PITCH	Thread pitch	Recommended for the following milling and drilling tool types: ■ **Tapping tools** ■ **Thread mill** ■ **Thread miller with chamfer** ■ **Thread mill with single thread** ■ **Thread mill w/ indexable insert** ■ **Thread drilling/milling cutter** ■ **Circular thread mill**
T-ANGLE	Point angle	Recommended for the following milling and drilling tool types: ■ **Drill** ■ **NC center drill** ■ **Countersink** ■ **Chamfer cutter**
NMAX	Maximum spindle speed	Optional
R_TIP	Radius at the tip	Recommended for the following milling and drilling tool types: ■ **Face mill** ■ **Countersink** ■ **Chamfer cutter**

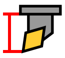
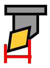
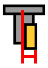
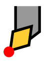
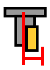
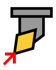
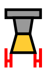
- All tool types listed in the **TYP** column are milling and drilling tools except for:
  - **Touch probe**
  - **Turning tool**
  - **Grinding wheel**
  - **Dressing tool**

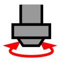
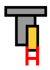
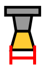
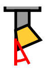
  Further information: "Tool types", Page 281
- The parameters are described in the tool table.

  Further information: "Tool table tool.t", Page 1995

Tool data for turning tools (option 50)

The control offers the following parameters for turning tools:

Icon and parameter	Meaning	Intended use
 **ZL**	Tool length 1	Required for all turning tool types
 **XL**	Tool length 2	Required for all turning tool types
 **YL**	Tool length 3	Required for all turning tool types
 **RS**	Cutting radius	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Recessing tool** ■ **Recess-turning tool**
 **TYPE**	Lathe tool type	Required for all turning tool types
 **TO**	Tool orientation	Required for all turning tool types Depending on the selected **TYPE** tool type, the control shows selected tool orientations with different graphics. The machine manufacturer can change this assignment.
 **DZL**	Delta value of tool length 1	Optional The control describes this value in connection with touch probe cycles.
 **DXL**	Delta value of tool length 2	Optional The control describes this value in connection with touch probe cycles.
 **DYL**	Delta value of tool length 3	Optional The control describes this value in connection with touch probe cycles.
 **DRS**	Delta value of cutter radius	Optional The control describes this value in connection with touch probe cycles.
 **DCW**	Delta value of cutter width	Optional The control describes this value in connection with touch probe cycles.

Icon and parameter	Meaning	Intended use
 **ORI**	Angle of orientation	Required for all turning tool types
 **T-ANGLE**	Tool angle	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Threading tool**
 **P-ANGLE**	Point angle	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Threading tool**
  **CUTLENGHT**	Cutting-edge length	Recommended
  **CUTWIDTH**	Tooth width	Required for the turning tool types below: ■ **Recessing tool** ■ **Recess-turning tool** Recommended for the other turning tool types
 **SPB-INSERT**	Angular offset	Required for all turning tool types

- The **TYP** column of the **Turning tool** tool type as well as the associated technology-specific tool types in the **TYPE** column define turning tools.

  Further information: "Tool types", Page 281

  Further information: "Types within the turning tools", Page 283
- The parameters are described in the turning tool table.

  Further information: "Turning tool table toolturn.trn (option 50)", Page 2006

Tool data for grinding tools (option 156)

NOTICE

Danger of collision!

In the tool management form, the control displays only the parameters relevant to the selected tool type. The tool tables contain locked parameters that are for internal consideration only. If you edit these additional parameters manually, tool data might no longer correctly match each other. There is a risk of collisions during subsequent movements!

- Edit the tools in the tool management form

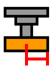
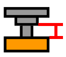
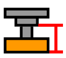
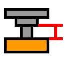
NOTICE

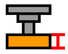
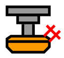
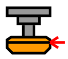
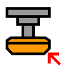
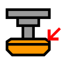
Danger of collision!

The control differentiates between freely editable and locked parameters. The control writes to the locked parameters and uses these parameters for internal consideration. You must not manipulate these parameters. If you manipulate the locked parameters, tool data might no longer correctly match each other. There is a risk of collisions during subsequent movements!

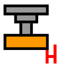
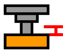
- Edit only freely editable tool management parameters
- Comply with the information about locked parameters in the tool data overview table

The control offers the following parameters for grinding tools:

Icon and parameter	Meaning	Intended use
 **TYPE**	Grinding tool type	Required for all grinding tool types
 **R-OVR**	Radius	Required for all grinding tool types This value must not be edited after initial dressing.
 **L-OVR**	Overhang	Required for the grinding tool types below: ■ **Conical grinding pin** ■ **Cup wheel** This value must not be edited after initial dressing.
 **LO**	Overall length	Required for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Conical grinding pin** This value must not be edited after initial dressing.
 **LI**	Length to the inner edge	Required for the **Conical grinding pin** grinding tool type This value must not be edited after initial dressing.

Icon and parameter	Meaning	Intended use
 **B**	Width	Required for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Cup wheel** This value must not be edited after initial dressing.
 **G**	Depth of grinding tool	Required for the **Cup wheel** grinding tool type This value must not be edited after initial dressing.
ALPHA	Slant angle	Required for the grinding tool types below: ■ **Conical grinding pin** ■ **Cup wheel** For the **Cup wheel** grinding tool type, you must define the angle 90°.
GAMMA	Corner angle	Required for the grinding tool types below: ■ **Conical grinding pin** ■ **Cup wheel**
 **RV**	Radius at the edge for **L-OVR**	Optional for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Conical grinding pin**
 **RV1**	Radius at the edge for **LO**	Optional for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Conical grinding pin**
 **RV2**	Radius at the edge for **LI**	Optional for the **Conical grinding pin** grinding tool type
 **HWI**	Angle for a relief cut on the inner edge	Required for the **Cup wheel** grinding tool type Optional for the remaining grinding tool types
 **HWA**	Angle for a relief cut on the outer edge	Required for the **Cup wheel** grinding tool type Optional for the remaining grinding tool types
COR_TYPE	Selection of compensation method	Required for all grinding tool types **Further information:** "Compensation methods", Page 253
INIT_D_OK	Initial dressing	Currently no function
MESS_OK	Measuring the grinding tool	The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**.
T-DRESS	Tool number of the dresser	The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Corresponds to parameter **A_NR_D** in the grinding tool table

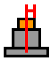
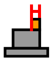
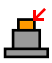
Icon and parameter	Meaning	Intended use
dR-OVR	Delta value of radius	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
dL-OVR	Delta value of overhang	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
dLO	Delta value of total length	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
dLI	Delta value of length up to the inner edge	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
DRESS-N-D	Default value of diameter dressing counter	Currently no function
DRESS-N-A	Default value of outer edge dressing counter	Currently no function Optional
DRESS-N-I	Default value of inner edge dressing counter	Currently no function Optional
DRESS-N-D-ACT	Diameter dressing counter	Currently no function
DRESS-N-A-ACT	Outer edge dressing counter	Currently no function
DRESS-N-I-ACT	Inner edge dressing counter	Currently no function
R_SHAFT	Radius of the tool shank	Optional
R_MIN	Min. permissible radius	Optional
B_MIN	Min. permissible width	Optional

Icon and parameter	Meaning	Intended use
 **V_MAX**	Maximum permissible cutting speed	Optional
 **AD**	Retraction amount at the diameter	Required for all grinding tool types
 **AA**	Retraction amount at the outer edge	Required for all grinding tool types
 **AI**	Retraction amount at the inner edge	Required for all grinding tool types

- The **TYP** column of the **Grinding wheel** tool type as well as the associated technology-specific tool types in the **TYPE** column define grinding tools.
  Further information: "Tool types", Page 281
  Further information: "Types within the grinding tools", Page 283
- The parameters are described in the grinding tool table.
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

Tool data for dressing tools (option 156)

The control offers the following parameters for dressing tools:

Icon and parameter	Meaning	Intended use
 **ZL**	Tool length 1	Required for dressing tool types
 **XL**	Tool length 2	Required for all dressing tool types
 **YL**	Tool length 3	Required for all dressing tool types
 **RS**	Cutting radius	Required for the dressing tool types below: ■ **Stationary dresser with radius** ■ **Rotating dresser with radius**
CUTWIDTH	Width of tooth	Required for the dressing tool types below: ■ **Stationary dresser (flat)** ■ **Rotating dresser (flat)**
 **TYPE**	Dressing tool type	Required for all dressing tool types
 **TO**	Tool orientation	Required for all dressing tool types
 **DZL**	Delta value of tool length 1	Optional
 **DXL**	Delta value of tool length 2	Optional
 **DYL**	Delta value of tool length 3	Optional
 **DRS**	Delta value of cutter radius	Optional
N-DRESS	Tool speed	Required for the dressing tool types below: ■ **Rotating dresser with radius** ■ **Rotating dresser (flat)**

- The **TYP** column of the **Dressing tool** tool type as well as the associated technology-specific tool types in the **TYPE** column define dressing tools.

  Further information: "Tool types", Page 281

  Further information: "Types within the dressing tools", Page 283
- The parameters are described in the dressing tool table.

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019

Tool data for touch probes

NOTICE

Danger of collision!

The control cannot use Dynamic Collision Monitoring (DCM) to protect L-shaped styluses against collisions. When using a touch probe with an L-shaped stylus there is a risk of collision!

- Carefully run in the NC program or program section in the **Program Run Single Block** operating mode
- Watch out for possible collisions!

The control offers the following parameters for touch probes:

Icon and parameter	Meaning	Intended use
L	Length	Required
R	Radius	Required
TP_NO	Number in the touch probe table	Required
TYPE	Type of touch probe	Required
F	Probing feed rate	Required
FMAX	Rapid traverse in probing cycle	Optional
F_PREPOS	Pre-positioning at rapid traverse	Required
TRACK	Orienting the touch probe in each probing process	Required When selecting **L-TYPE** in the **STYLUS** parameter, **ON** must be selected
REACTION	Trigger **NCSTOP** or **EMERGSTOP** in case of collision	Required
SET_UP	Set-up clearance	Recommended

Icon and parameter	Meaning	Intended use
DIST	Maximum measuring range	Recommended
CAL_OF1	Center offset in the main axis	Required when **ON** is selected in parameter **TRACK** The control describes this value in connection with the calibration cycle.
CAL_OF2	Center offset in the secondary axis	Required when **ON** is selected in parameter **TRACK** The control describes this value in connection with the calibration cycle.
CAL_ANG	Spindle angle during calibration	Required when **ON** is selected in parameter **TRACK**
STYLUS	Shape of the stylus	Required If you do not define the parameter, the control uses **SIMPLE**

- The **TYP** column of the **Touch probe** tool type as well as the touch probe model in the **TYPE** column define touch probes.

  Further information: "Tool types", Page 281
- The parameters are described in the touch probe table.

  Further information: "Touch probe table tchprobe.tp", Page 2022

11.4 Tool management

Application

The control displays the tool definitions of all technologies as well as the tools currently present in the tool magazine in the **Tool management** application of the **Tables** operating mode.

The tool management allows adding tools, editing tool data and deleting tools.

Related topics

- Creating new tools
  Further information: "Configuring a tool", Page 152
- Table workspace
  Further information: "Table workspace", Page 1982
- Form workspace
  Further information: "Form workspace for tables", Page 1989

Description of function

You can define up to 32,767 tools in the tool management; this is the maximum number of available table rows.

The control displays all tool data of the tool tables below in the tool management:

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 1995
- Turning tool table **toolturn.trn** (option 50)
  Further information: "Turning tool table toolturn.trn (option 50)", Page 2006
- Grinding tool table **toolgrind.grd** (option 156)
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010
- Dressing tool table **tooldress.drs** (option 156)
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019
- Touch probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 2022

The control additionally displays the pockets occupied in the magazine from pocket table **tool_p.tch** in the tool management.

Further information: "Pocket table tool_p.tch", Page 2026

Tool data can be edited in the **Table** workspace or in the **Form** workspace. In the **Form** workspace the control shows the correct tool data for each tool type.

Further information: "Tool data", Page 275

Notes

- When creating a new tool, the length **L** and radius **R** columns are empty at first. The control will not insert a tool whose length and radius are missing and will display an error message.
- The tool data of tools still stored in the pocket table cannot be deleted. The tools must be removed from the magazine first.
- When editing tool data, bear in mind that the current tool may have been entered in column **RT** as a replacement tool of another tool!
- If the cursor is within the **Table** workspace and the **Edit** toggle switch is deactivated, a search using the keyboard can be started. The control opens a separate window with an input field and automatically searches for the entered string. If it finds a tool with the entered characters, the control selects this tool. If it finds several tools with this string of characters, you can scroll up and down in the window.

11.4.1 Importing and exporting tool data

Application

The control can import and export tool data. This avoids manual editing efforts and possible typing errors. Importing tool data is particularly useful in connection with a tool presetter. Exported tool data can be used for the tool database of your CAM system, for example.

Description of function

The control transmits tool data as a CSV file.

Further information: "File types", Page 1149

The tool data transfer file is structured as follows:

- The first row contains the tool table column names that are transferred.
- The other rows contain the tool data to be transferred. The order of the data must match the order of the column names in row 1. A period is used as decimal separator.

The column names and the tool data stand between double quotation marks and are separated by semicolons.

Please note the following regarding the transfer file:

- The tool number must be present.
- Any tool data can be imported. The data record does not need to contain all tool table column names or all tool data.
- Missing tool data contain no value between the quotation marks.
- The column names can be arranged in any order. The order of tool data must match the order of column names.

Importing tool data

To import tool data:

- Select the **Tables** operating mode
- Select **Tool management**

- Activate **Edit**
- > The control enables tool management for editing.

- Select **Import**
- > The control opens a selection window.
- Select the desired CSV file

- Select **Import**
- > The control adads the tool data to the tool management.
- > If required, the control opens the **Confirm import** window (e.g., in case of identical tool numbers).
- Selecting the procedure:
  - **Append**: the control adds the tool data as new rows at the end of the table.
  - **Overwrite**: the control overwrites the initial tool data with the tool data from the transfer file.
  - **Cancel**: the control cancels the import process.

NOTICE

Caution: Data may be lost!

When overwriting existing tool data with the **Overwrite** function, the control will permanently delete the initial tool data!

- Use this function only with tool data that are no longer needed

Exporting tool data

To export tool data:

- Select the **Tables** operating mode
- Select **Tool management**
- Activate **Edit**
- > The control enables tool management for editing.
- Mark the tool to be exported
- Open the context menu with a long press or by right-clicking

  Further information: "Context menu", Page 1522
- Select **Mark row**
- Mark further tools if required
- Select **Export**
- > The control opens the **Save as** window.
- Select a path

> By default, the control saves the transfer file under **TNC:\table**.

- Enter the file name
- Select the file type

> Select either **TNC7 (*.csv)** or **TNC 640 (*.csv)**. The internal formatting of the transfer files differs. If you wish to use the data in a previous control model, select **TNC 640 (*.csv)**.

- Select **Create**
- > The control will save the file using the selected path.

Notes

NOTICE

Caution: Possible material damage!

If the transfer file contains unknown column names, the control will not accept the data from this column! In this case, the control will perform the operations with an incompletely defined tool.

- Check whether the column names are correct
- After importing, check the tool data and adapt them if required.

- The transfer file must be saved under **TNC:\table**.
- The internal formatting of the transfer files differs:
  - **TNC7 (*.csv)** writes the values between double quotation marks and separates the values by semicolons
  - **TNC 640 (*.csv)** writes the values partly between brackets and separates the values by commas

  The TNC7 can both import and export both transfer files.

11.5 Tool carrier management

Application

Tool carrier management allows parameterizing and assigning tool carriers.

The control represents the tool carriers graphically in the simulation and takes the tool carriers into account by calculation, e. g. in Dynamic Collision Monitoring (DCM, option 40).

Related topics

- **Simulation** workspace

  Further information: "Simulation Workspace", Page 1535
- Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164

Description of function

To ensure that the control takes the tool carriers into account in its calculations and in the display:

- Save the tool carrier or tool carrier templates
- Parameterize the tool carrier templates

  Further information: "Parameterizing tool carrier templates", Page 303
- Assign a tool carrier

  Further information: "Assigning a tool carrier", Page 303

> If you are using M3D or STL files instead of tool carrier templates, you can assign the files directly to the tools. The parameterization step is superfluous here.
>
> Tool carriers in STL format must meet the following requirements:
>
> - Max. 20 000 triangles
> - Triangular mesh forms a closed shell
>
> If an STL file does not meet the requirements of the control, then the control issues an error message.
>
> For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.
>
> **Further information:** "Options for fixture files", Page 1172

Tool-carrier templates

Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. HEIDENHAIN provides ready-to-use tool carrier templates for downloading. Tool carrier templates are 3D models with fixed geometries but changeable dimensions.

Tool carrier templates must be stored as files with a ***.cft** file name extension in the **TNC:\system\Toolkinematics** directory.

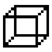
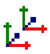
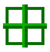
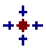
> They can be downloaded through the following link:
>
> **http://www.klartext-portal.com/nc-solutions/en**
>
> If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

The tool carrier templates can be parametrized in the **ToolHolderWizard** window. This defines the tool carrier dimensions.

Further information: "Parameterizing tool carrier templates", Page 303

The parametrized tool carriers with the ***.cfx** extension are saved at **TNC:\system \Toolkinematics**.

The **ToolHolderWizard** window contains the following icons:

Icon	Function
	Close the application
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Display or hide transformation vectors
	Show or hide names of collision objects
	Display or hide test points
	Show or hide measurement points
	Restore initial view
	Select alignment, e.g. plan view

11.5.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:

- Select the **Files** operating mode
- Open the **TNC:\system\Toolkinematics** folder
- Double-tap or -click desired tool carrier template with the ***.cft** extension
- > The control opens the **ToolHolderWizard** window.
- Define the dimensions in the **Parameter** area
- Define a name with the ***.cfx** extension in the **Output file** area
- Select **Generate file**
- > The control shows the message that the tool carrier template was successfully generated and saves the file in the folder **TNC:\system\Toolkinematics**.
- Select **OK**
- Select **Quit**

11.5.2 Assigning a tool carrier

To assign a tool carrier to a tool:

- Select the **Tables** operating mode
- Select **Tool management**
- Select the tool you want to use
- Activate **Edit**
- In the **Spec. functions** panel, select the **KINEMATIC** parameter
- > The control displays the available tool carriers in the **Tool-carrier kinematics** window.
- Select the desired tool carrier
- Select **OK**
- > The control assigns the tool carrier to the tool.

> - The tool carrier will only be taken into account after the next tool call.
> - Parameterized tool carriers can consist of several subfiles. If the subfiles are incomplete, the control will display an error message.
>
>   Only use fully parameterized tool carriers and error-free STL or M3D files!
>
>   For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.
>
>   **Further information:** "Fixture monitoring (option 40)", Page 1171

Notes

- In the simulation, the tool carriers can be checked for collisions with the workpiece.

  Further information: "Advanced checks in the simulation", Page 1190
- On 3-axis machines with rectangular angle heads, tool carriers of angle heads are advantageous in connection with the tool axes **X** and **Y** because the control takes the dimensions of the angle heads into account.

  HEIDENHAIN recommends machining with tool axis **Z**. Using Advanced Functions Set 1 (software option 8), you can tilt the working plane to the angle of exchangeable angle heads and continue working with tool axis **Z**.
- The control monitors the tool carriers by means of Dynamic Collision Monitoring (DCM, option 40). This enables the tool carriers to protect against collisions with fixtures or machine components.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
- A grinding tool that is to be dressed must not contain any tool carrier kinematics (option 156).

11.6 Tool call

11.6.1 Tool call by TOOL CALL

Application

The **TOOL CALL** function calls a tool in the NC program. When the tool is in the tool magazine, the control inserts the tool into the spindle. When the tool is not in the magazine, you can insert it by hand.

Related topics

- Automatic tool change with **M101**

  Further information: "Automatically inserting a replacement tool with M101", Page 1355
- Tool table **tool.t**

  Further information: "Tool table tool.t", Page 1995
- Pocket table **tool_p.tch**

  Further information: "Pocket table tool_p.tch", Page 2026

Requirement

- Tool defined

  To call a tool, the tool must be defined in the tool management.

  Further information: "Tool management ", Page 297

Description of function

Upon calling a tool, the control reads the associated row from the tool management. The tool data are visible on the **Tool** tab of the **Status** workspace.

Further information: "Tool tab", Page 184

> HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
>
> **Further information:** "Overview of miscellaneous functions", Page 1319

Icons

The NC function **TOOL CALL** offers the following icons:

Icon or shortcut	Function
	Open selection window for tools
	In the application **Tool management**, switch to the selected tool You can change the tool as needed. **Further information:** "Tool management ", Page 297
	Open the **Cutting data calculator** **Further information:** "Cutting data calculator", Page 1529

Input

11 TOOL CALL 4 .1 Z S10000 F750 DL +0,2 DR+0,2 DR2+0,2	; Call the tool

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL CALL	Syntax initiator for a tool call
4, **QS4** or **"MILL_D8_ROUGH"**	Tool definition as a fixed or variable number or name Only the tool definition as a number is unique because the tool names of several tools may be identical! Syntax element depending on technology or application Selection by means of a selection window **Further information:** "Technology-dependent differences when calling tools", Page 307
.1	Step index of the tool Optional syntax element **Further information:** "Input", Page 306
Z	Tool axis By default, tool axis **Z**. Other possibilities might be available, depending on the machine. Syntax element depending on technology or application **Further information:** "Technology-dependent differences when calling tools", Page 307
S or **S(VC =)**	Spindle speed or cutting speed Optional syntax element **Further information:** "Spindle speed S", Page 309
F, **FZ** or **FU**	Feed rate Alternative feed specifications: feed per tooth or feed per revolution Optional syntax element **Further information:** "Feed rate F", Page 310
DL	Delta value of tool length Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 1112
DR	Delta value of the tool radius Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 1112
DR2	Delta value of the tool radius 2 Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 1112

Technology-dependent differences when calling tools

Milling cutter tool call

The following tool data of a milling cutter can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Tool axis
- Spindle speed
- Feed rate
- DL
- DR
- DR2

Calling a milling cutter requires the number or the name of the tool, the tool axis and the spindle speed.

Further information: "Tool table tool.t", Page 1995

Tool call of a turning tool (option 50)

The following tool data of a turning tool can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Feed rate

Calling a turning tool requires the number or the name of the tool.

Further information: "Turning tool table toolturn.trn (option 50)", Page 2006

Tool call of a grinding tool (option 156)

The following tool data of a grinding tool can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Tool axis
- Spindle speed
- Feed rate

Calling a grinding tool requires the number or the name of the tool and the tool axis.

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

Tool call of a dressing tool (option 156)

The following tool data of a dressing tool can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Feed rate

Calling a dressing tool requires the number or the name of the tool!

Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019

A dressing tool can be called only in dressing mode!

Further information: "Activating dressing mode with FUNCTION DRESS", Page 254

The dressing tool will not be mounted to the spindle. You need to mount the dressing tool manually to a pocket defined by the machine manufacturer. Additionally, you must define the tool in the pocket table.

Further information: "Pocket table tool_p.tch", Page 2026

Tool call of a workpiece touch probe (option 17)

The following tool data of a workpiece touch probe can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Tool axis

Calling a workpiece touch probe requires the number or the name of the tool and the tool axis!

Further information: "Touch probe table tchprobe.tp", Page 2022

Updating tool data

A **TOOL CALL** allows updating the data of the active tool even without tool change (e.g., modifying the cutting data or delta values). The tool data that can be modified depend on the technology.

In the cases below, the control updates only the data of the active tool:

- Without tool number or tool name and without tool axis
- Without tool number or tool name and with the same tool axis as in the previous tool call

When a tool number or a tool name or a changed tool axis is programmed in tool call, the control runs a tool change macro.

This may cause the control to insert a replacement tool because the service life has expired.

Further information: "Automatically inserting a replacement tool with M101", Page 1355

Notes

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 machine manufacturer uses the machine parameter **allowToolDefCall** (no. 118705) to specify whether a tool can be defined by its name, its number or both in the **TOOL CALL** and **TOOL DEF** functions.

  Further information: "Tool pre-selection by TOOL DEF", Page 311
- The machine manufacturer uses the optional machine parameter **progToolCallDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

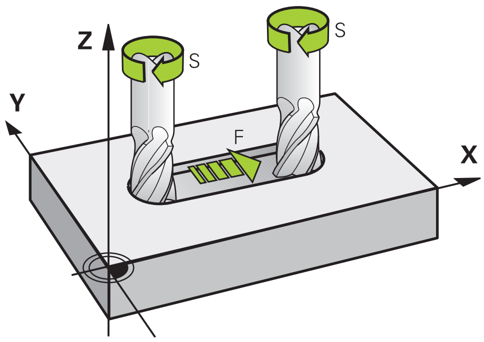
  Further information: "Tool compensation for tool length and tool radius", Page 1112

  Further information: "Positions workspace", Page 163

11.6.2 Cutting data

Application

The cutting data consist of spindle speed **S** or alternatively constant cutting speed **VC** and feed rate **F**.



Description of function

Spindle speed S

The spindle speed **S** can be defined in the following ways:

- Tool call with **TOOL CALL**
  Further information: "Tool call by TOOL CALL", Page 304
- Button **S** of the **Manual operation** application
  Further information: "Manual operation application", Page 202

The spindle speed **S** is defined as spindle revolutions per minute (rpm).

Alternatively, the constant cutting speed **VC** in meters per minute (m/min) can be defined.

Further information: "Technology values for turning operations", Page 238

Effect

The spindle speed or the cutting speed is active until a new spindle speed or cutting speed is defined in a **TOOL CALL** NC block.

Potentiometers

The speed potentiometer allows varying the spindle speed between 0% and 150% while the program is running. The speed potentiometer setting is active only for machines with infinitely variable spindle drive. The maximum spindle speed depends on the machine.

Further information: "Potentiometers", Page 122

Status displays

The control displays the current spindle speed in the following workspaces:

- **Positions** workspace
  Further information: "Positions workspace", Page 163
- **POS** tab of the **Status** workspace
  Further information: "POS tab", Page 179

Feed rate F

The feed rate **F** can be defined in the following ways:

- Tool call with **TOOL CALL**
  Further information: "Tool call by TOOL CALL", Page 304
- Positioning block
  Further information: "Path Functions", Page 317
- Button **F** of the **Manual operation** application
  Further information: "Manual operation application", Page 202

The feed rate for linear axes is defined in millimeters per minute (mm/min).

The feed rate for rotary axes is defined in degrees per minute (°/min).

The feed rate can be defined with an accuracy of three decimal places.

Alternatively, the feed rate can be defined in the NC program or in a tool call in the following units:

- Feed rate per tooth **FZ** in mm/tooth
  FZ defines the path in millimeters that the tool covers per tooth.

> When using **FZ**, the number of teeth must be defined in the **CUT** column of the tool management.
>
> **Further information:** "Tool management ", Page 297

- Feed rate per revolution **FU** in mm/rev
  FU defines the path in millimeters that the tool covers per spindle revolution.
  The feed rate per revolution is used mainly for turning (option 50).
  Further information: "Feed rate", Page 240

The feed rate defined in a **TOOL CALL** can be called up within the NC program, using **F AUTO**.

Further information: "F AUTO", Page 310

The feed rate defined in the NC program is active up to the NC block in which a new feed rate is programmed.

F MAX

If you define **F MAX**, the control moves at rapid traverse. **F MAX** is non-modal, i.e., it is active only in the block where it is called. Starting with the subsequent NC block, the last previously defined feed rate is active again. The maximum feed rate depends on the machine and may depend on the axis.

Further information: "Feed rate limit FMAX", Page 1958

F AUTO

If you defined a feed rate in a **TOOL CALL** block, this feed rate can be used in the next positioning blocks, using **F AUTO**.

Button F of the Manual operation application

- If you enter F=0, then the feed rate that the machine manufacturer has defined as minimum feed rate is active
- If the feed rate you entered exceeds the maximum value that has been defined by the machine manufacturer, then the value defined by the machine manufacturer is active
  Further information: "Manual operation application", Page 202

Potentiometers

The feed-rate potentiometer allows varying the feed rate between 0% and 150% while the program is running. The setting of the feed-rate potentiometer is active only for the programmed feed rate. As long as the programmed feed rate has not yet been reached, the feed-rate potentiometer has no effect.

Further information: "Potentiometers", Page 122

Status displays

The control displays the current feed rate in mm/min in the following workspaces:

- **Positions** workspace

  Further information: "Positions workspace", Page 163
- **POS** tab of the **Status** workspace

  > In the **Manual operation** application, the control displays the feed rate with decimal places on the **POS** tab. The control displays the feed rate with a total of six decimal places.

  Further information: "POS tab", Page 179
- The control displays the contouring feed rate as follows:
  - If **3D ROT** is active, the contouring feed rate is displayed if multiple axes are moving
  - If **3D ROT** is inactive, the feed-rate display remains empty when more than one axis is moved simultaneously
  - If a handwheel is active, the control shows the contouring feed rate during program run.

  Further information: "3-D rotation window (option 8)", Page 1098

Notes

- In inch programs, the feed rate must be defined in 1/10 inch/min.
- 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.
- When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.

11.6.3 Tool pre-selection by TOOL DEF

Application

Using **TOOL DEF**, the control prepares a tool in the magazine, thus reducing the tool change time.

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

Description of function

If your machine is equipped with a chaotic tool changer system and a double gripper, you can perform tool pre-selection. To do this, program the **TOOL DEF** function after a **TOOL CALL** data record and select the tool to be used next in the NC program. The control prepares the tool while the program is running.

Input

11 TOOL DEF 2 .1	; Tool pre-selection

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL DEF	Syntax initiator for tool pre-selection
2, **QS2** or **"MILL_D4_ROUGH"**	Tool definition as a fixed or variable number or name Only the tool definition as a number is unique because the tool names of several tools may be identical!
.1	Step index of the tool **Further information:** "Indexed tool", Page 276 Optional syntax element

This function can be used for all technologies except for dressing tools (option 156).

Application example

11 TOOL CALL 5 Z S2000	; Call the tool
12 TOOL DEF 7	; Pre-select the next tool
*** - ...**	
21 TOOL CALL 7	; Call the pre-selected tool

11.7 Tool usage test

Application

The tool usage test allows checking the tools used in the NC program before starting the program. The control checks if the tools used are available in the machine magazine and have sufficient remaining tool life. Any missing tools can be stored in the machine or tools can be exchanged due to insufficient remaining tool life before starting the program. This avoids interruptions while the program is running.

Related topics

- Contents of the tool usage file
  Further information: "Tool usage file", Page 2029
- Tool usage test in Batch Process Manager (option 154)
  Further information: "Batch Process Manager (option 154)", Page 1943

Requirement

- A tool usage file is needed for performing a tool usage test
  The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.
  Further information: "Tool usage file", Page 2029
- The **Generate tool-usage file** function setting is set to either **once** or **always**
  Further information: "Channel settings", Page 2104
- Use the same tool table for the simulation as for the program run
  Further information: "Simulation Workspace", Page 1535

Description of function

Creating the tool usage file

A tool usage file must be generated for performing the tool usage test.

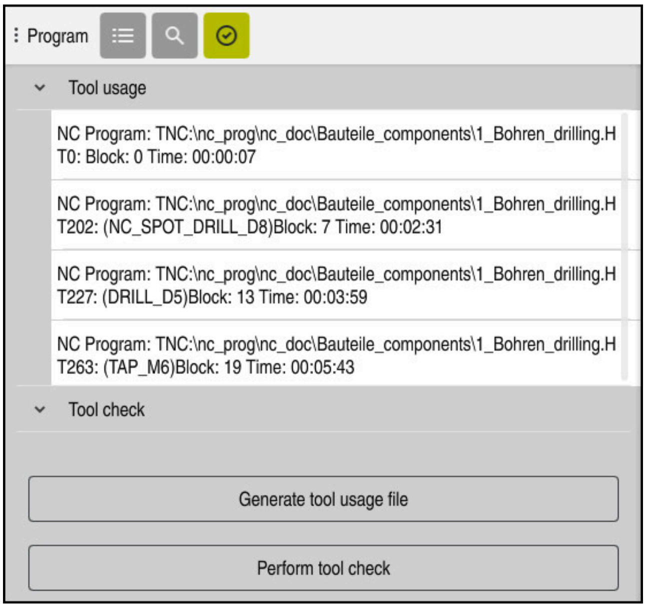
When setting the **Generate tool-usage file** setting to **once** or **always**, the control will generate a tool usage file in the following cases:

- Simulating the NC program completely
- Executing the NC program completely
- Select **Generate tool usage file** in the **Tool check** column of the **Program** workspace

The control saves the tool usage file with the ***.t.dep** extension in the same folder where the NC program is stored.

Further information: "Tool usage file", Page 2029

Tool check column in the Program workspace



Tool check column in the **Program** workspace

In the **Tool check** column of the **Program** workspace, the control displays the following areas:

- **Tool usage**
  Further information: "Tool usage area", Page 314
- **Tool check**
  Further information: "Tool check area", Page 314

Further information: "Program workspace", Page 217

Tool usage area

The **Tool usage** area is empty before generating a tool usage file.

Further information: "Creating the tool usage file", Page 313

Further information: "Tool usage file", Page 2029

The control displays the chronological order of all tool calls in the **Tool usage** area, along with the following information:

- Path of NC program in which the tool is called
- Tool number and possibly tool name
- Row number of tool call in NC program
- Tool usage time between the tool changes

Tool check area

Before performing a tool usage test with the **Tool check** button, the **Tool check** area has no content.

Further information: "Performing the tool usage test", Page 315

When performing the tool usage test, the control checks the following:

- The tool is defined in the tool management

  Further information: "Tool management ", Page 297
- The tool is defined in the pocket table

  Further information: "Pocket table tool_p.tch", Page 2026
- The tool has sufficient remaining tool life

  The control checks if the remaining tool life **TIME1** minus **CUR_TIME** is sufficient for the machining process. To meet this requirement, the remaining tool life must be longer than the tool usage time **WTIME** from the tool usage file.

  Further information: "Tool table tool.t", Page 1995

  Further information: "Tool usage file", Page 2029

The control displays the following information in the **Tool check** area:

- **OK**: All tools are available and have sufficient remaining tool life
- **No suitable tool**: The tool is not defined in the tool management

  In this case, check if the correct tool is selected in the tool call. Otherwise, create the tool in the tool management.
- **External tool**: The tool is defined in the tool management, but not in the pocket table

  If your machine is equipped with a magazine, position the missing tool in the magazine.
- **Insufficient remaining tool life**: The tool is blocked or does not have sufficient remaining tool life

  Change the tool or use a replacement tool.

  Further information: "Tool call by TOOL CALL", Page 304

  Further information: "Automatically inserting a replacement tool with M101", Page 1355

> If you double-tap or click a tool entry in the **Tool usage** or **Tool check** areas, the control switches to the tool selected in tool management. You can make modifications as needed.

11.7.1 Performing the tool usage test

Use the tool usage test as follows:

- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Machine settings** group
- Select the **Machine settings** menu item
- In the **Channel settings** area, select Generate tool usage file **Once** for the simulation.

  Further information: "Channel settings", Page 2104
- Press **Apply**
- Select the **Editor** operating mode
- Select **Add**
- Select the desired NC program
- Select **Open**
- > The control opens the NC program in a new tab.
- Select the **Tool check** column
- > The control opens the **Tool check** column.
- Select **Generate tool usage file**
- > The control generates a tool usage file and displays the tools used in the **Tool usage** area.

  Further information: "Tool usage file", Page 2029
- Select **Perform tool check**
- > The control performs the tool usage test.
- > The **Tool check** area of the control shows whether all tools are available and have sufficient remaining tool life.

Notes

- If you selected **never** in the **Generate tool-usage file** function, the **Generate tool usage file** button in the **Tool check** column is grayed out.

  Further information: "Channel settings", Page 2104
- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.

  Further information: "Simulation Workspace", Page 1535
- The control creates dependency files (***.dep**); for example, the tool-usage file in order to perform a tool usage test.

  Further information: "Tool usage file", Page 2029
- The control displays the order of tool calls of the NC program that is active in the program run in the **T usage order** table (option 93).

  Further information: "T usage order (option 93)", Page 2031
- The control shows an overview of all tool calls of the NC program that are active in the program run in the **Tooling list** table (option 93).

  Further information: "Tooling list (option 93)", Page 2033
- Function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.
- Function **FN 18: SYSREAD ID975 NR2 IDX** allows querying the tool usage test for a pallet table. The pallet table row is defined by **IDX**.
- The machine manufacturer uses the machine parameter **autoCheckPrg** (no. 129801) to define whether the control automatically generates a tool usage file upon selecting an NC program.
- The machine manufacturer uses the machine parameter **autoCheckPal** (no. 129802) to define whether the control automatically generates a tool usage file upon selecting a pallet table.
- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files with the *.dep extension in the file manager. Even if the control displays no dependency files, it still generates a tool usage file.

12

Path Functions

12.1 Fundamentals of coordinate definitions

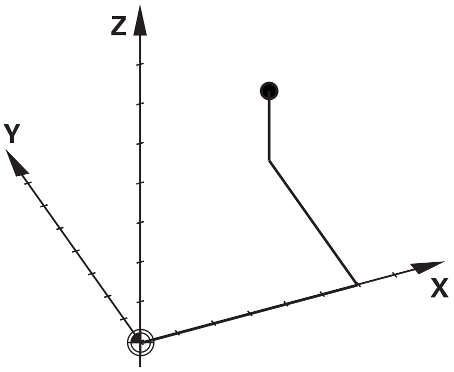
You program a workpiece by defining the path contours and the target coordinates.

Depending on the dimensioning used in the technical drawing, you use Cartesian or polar coordinates with absolute or incremental values.

12.1.1 Cartesian coordinates

Application

A Cartesian coordinate system consists of two or three axes that are all mutually perpendicular. Cartesian coordinates are relative to the datum (origin) of the coordinate system, which is at the intersection of the axes.



With Cartesian coordinates you can uniquely specify a point in space by defining the three axis values.

Description of function

In the NC program you define the values in the linear axes **X**, **Y**, and **Z**, such as with a straight line **L**.

```
11 L  X+60 Y+50 Z+20  RL F200
```

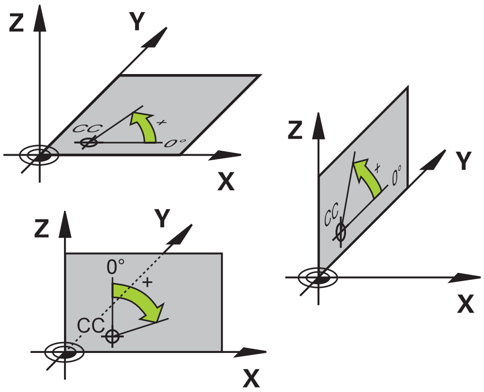
The programmed coordinates are modally effective. As long as the value of an axis remains the same, you do not need to program the value for further path contours.

12.1.2 Polar coordinates

Application

You define polar coordinates in one of the three planes of a Cartesian coordinate system.

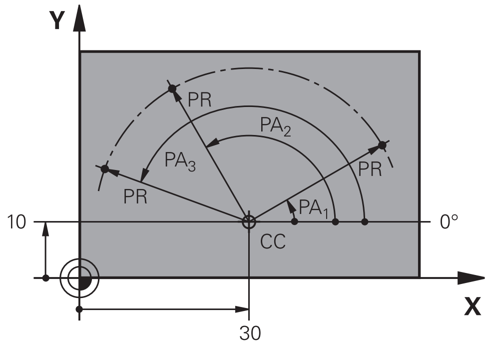
Polar coordinates are relative to a previously defined pole. From this pole you define a point by its distance to the pole and the angle to the angle reference axis.



Description of function

Polar coordinates can be used in, for example, the following situations:

- Points on circular paths
- Workpiece drawings with angular information, such as bolt hole circles



You define the pole **CC** with Cartesian coordinates in two axes. These axes specify the plane and the angle reference axis.

The pole is modally effective within an NC program.

The angle reference axis is related to the plane as follows:

Plane	Angle reference axis
XY	+X
YZ	+Y
ZX	+Z

```
11 CC  X+30 Y+10
```

The polar coordinate radius **PR** is relative to the pole. **PR** defines the distance of this point from the pole.

The polar coordinate angle **PA** defines the angle between the angle reference axis and this point.

```
11 LP  PR+30 PA+10  RR F300
```

The programmed coordinates are modally effective. As long as the value of an axis remains the same, you do not need to program the value for further path contours.

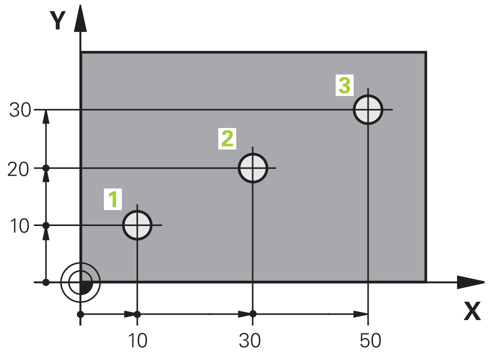
12.1.3 Absolute input

Application

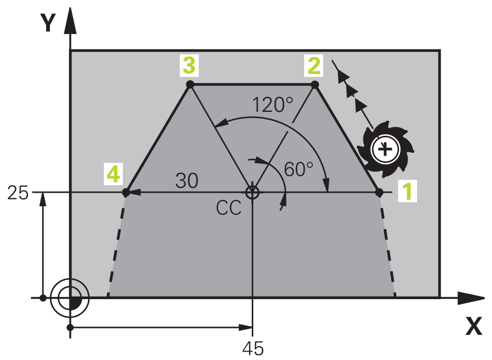
Absolute input always references an origin. For Cartesian coordinates, the origin is the datum, and for polar coordinates the origin is the pole and the angle reference axis.

Description of function

Absolute values define the target point for positioning.



11 L X+10 Y+10 RL F200 M3	; Position at point 1
12 L X+30 Y+20	; Position at point 2
13 L X+50 Y+30	; Position at point 3



11 CC X+45 Y+25	; Define the pole with two axes using Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position at point 1
13 LP PA+60	; Position at point 2
14 LP PA+120	; Position at point 3
15 LP PA+180	; Position at point 4

12.1.4 Incremental entries

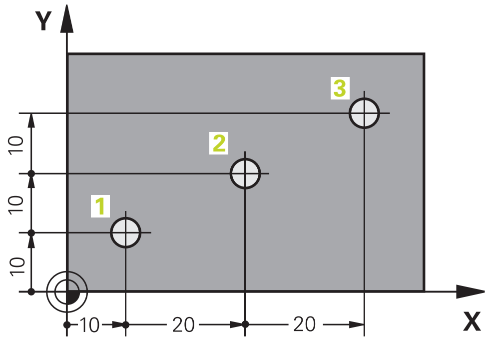
Application

Incremental entries are always referenced to the previously programmed coordinates. For Cartesian coordinates those are the values in the axes **X**, **Y**, and **Z**, and for polar coordinates the value of the polar coordinate radius **PR** and the polar coordinate angle **PA**.

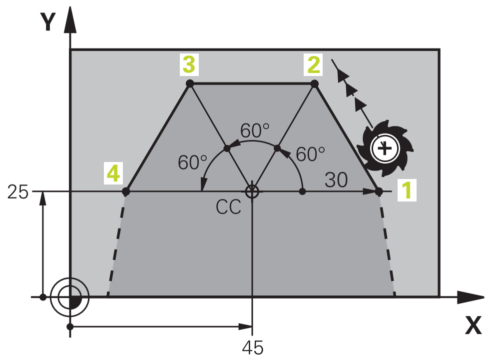
Description of function

Incremental entries define the value by which the control positions. The previously programmed coordinates serve as the respective datum of the coordinate system.

You define incremental coordinates with an **I** before each axis designation.



11 L X+10 Y+10 RL F200 M3	; Position to point 1 absolutely
12 L IX+20 IY+10	; Position to point 2 incrementally
13 L IX+20 IY+10	; Position to point 3 incrementally



11 CC X+45 Y+25	; Define the pole absolutely in two axes with Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position to point 1 absolutely
13 LP IPA+60	; Position to point 2 incrementally
14 LP IPA+60	; Position to point 3 incrementally
15 LP IPA+60	; Position to point 4 incrementally

12.2 Fundamentals of path functions

Application

When creating an NC program, you can use the path functions to program the individual contour elements. To do so, use coordinates to define the end points of the contour elements.

The control then uses the coordinate entries, the tool data, and the radius compensation to calculate the traverse path. The control simultaneously positions all machine axes that you programmed in the NC block of a path function.

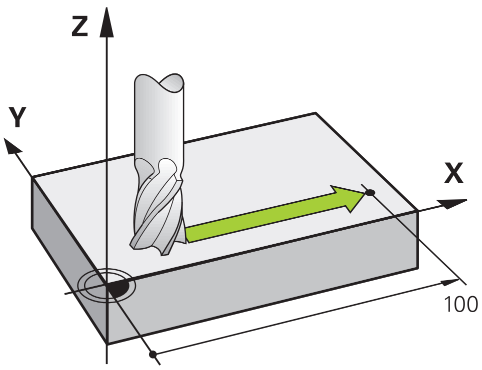
Description of function

Inserting a path function

The gray path function keys initiate the dialog. The control inserts the NC block in the NC program and prompts you for each piece of necessary information.

ℹ	Depending on the design of the machine tool, either the tool moves or the machine table moves. When programming a path function, you always assume that the tool is in motion.

Motion in one axis



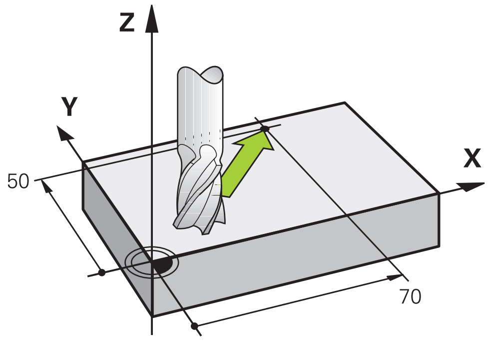
If the NC block contains one coordinate, the control moves the tool parallel to the programmed machine axis.

Example

```
L  X+100
```

The tool retains the Y and Z coordinates and moves to the position **X+100**.

Motion in two axes



If the NC block contains two coordinates, the control moves the tool in the programmed plane.

Example

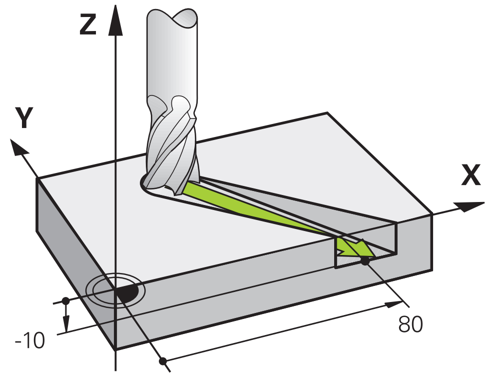
```
L X+70  Y+50
```

The tool retains the Z coordinate and moves in the XY plane to the position **X+70 Y+50**.

You define the working plane by entering the tool axis when calling the tool with **TOOL CALL**.

Further information: "Designation of the axes on milling machines", Page 208

Motion in more than two axes



If the NC block contains three coordinate entries, the control moves the tool spatially to the programmed position.

Example

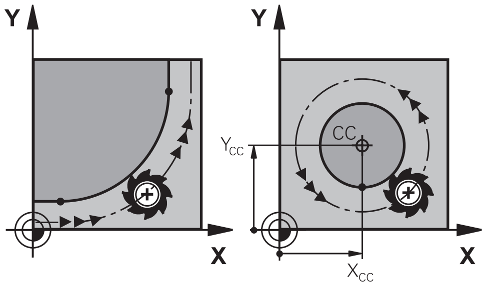
```
L X+80  Y+0 Z-10
```

Depending on the kinematics of your machine, you can program up to six axes in a linear **L** block.

Example

```
L X+80 Y+0 Z-10 A+15 B+0 C-45
```

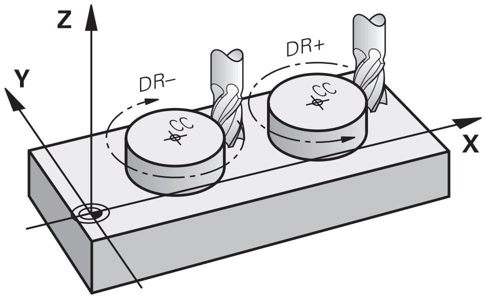
Circles and arcs



Use the path functions for circular arcs to program circular motions in the working plane.

The control moves the tool in two axes simultaneously on a circular path relative to the workpiece. You can program circular paths with a circle center point **CC**.

Direction of rotation DR for circular motions



When a circular path has no tangential transition to another contour element, define the direction of rotation as follows:

- Clockwise direction of rotation: **DR-**
- Counterclockwise direction of rotation: **DR+**

Tool radius compensation

Tool radius compensation is defined in the NC block of the first contour element.

Do not activate tool radius compensation in an NC block for a circular path. Activate tool radius compensation in a preceding straight line.

Further information: "Tool radius compensation", Page 1114

Pre-positioning

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

12.3 Path functions with Cartesian coordinates

12.3.1 Overview of path functions

Key	Function	Further information
	Straight line **L** (line)	Page 326
	Chamfer **CHF** (chamfer) Chamfer between two straight lines	Page 328
	Rounding **RND** (rounding of corner) Circular arc with tangential connection to the preceding and subsequent contour elements	Page 330
	Circle center point **CC** (circle center)	Page 332
	Circular path **C** (circle) Circular path around a circle center **CC** to an end point	Page 334
	Circular path **CR** (circle by radius) Circular path with a specified radius	Page 336
	Circular path **CT** (circle tangential) Circular path with tangential connection to the preceding contour element	Page 339

12.3.2 Straight line L

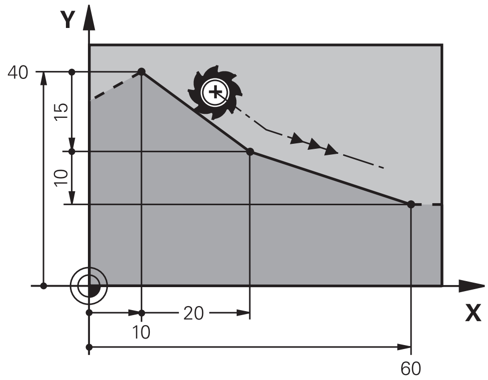
Application

With a straight line **L** you program a straight traverse motion in any direction.

Related topics

- Programming a straight line with polar coordinates
  Further information: "Straight line LP", Page 346

Description of function



The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

Depending on the kinematics of your machine, you can program up to six axes in a linear **L** block.

Input

11 L X+50 Y+50 R0 FMAX M3	; Straight line without radius compensation in rapid traverse

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **L**

The NC function includes the following syntax elements:

Syntax element	Meaning
L	Syntax initiator for a straight line
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	End point of the straight line as a fixed or variable number Entry: absolute or incremental Optional syntax element
&X, **&Y**, **&Z**	End point of the straight line in a main axis that is deselected with **PARAXMODE** as a fixed or variable number **Further information:** "Select three linear axes for machining with FUNCTION PARAXMODE", Page 1289 Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Notes

- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.

  Further information: "Form column in the Program workspace", Page 227
- The **actual position capture** key allows you to program a straight line **L** with all axis values. The values are equivalent to the **Actual pos. (ACT)** mode of the position display.

  Further information: "Position displays", Page 188

Example

```
11 L  Z+100  R0 FMAX  M3
12 L  X+10 Y+40  RL F200
13 L  IX+20 IY-15
14 L  X+60 IY-10
```

12.3.3 Chamfer CHF

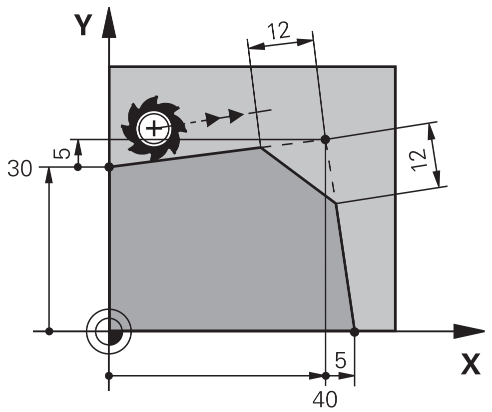
Application

The **CHF** chamfer function allows you to insert a chamfer between two straight lines. The size of the chamfer is based on the intersection that you have programmed with the straight lines.

Requirements

- Straight lines in the working plane before and after the chamfer
- Identical tool compensation before and after the chamfer
- Chamfer is machinable with the current tool

Description of function



Cutting two straight lines creates contour corners. You can insert a chamfer at these contour corners. The angle of the corner is irrelevant; you simply define the length by which each straight line is shortened. The control does not traverse to the corner point.

If you program a feed rate in the **CHF** block, then this feed rate is in effect only while cutting the chamfer.

Input

11 CHF 1 F200	; Chamfer with a size of 1 mm

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **CHF**

The NC function includes the following syntax elements:

Syntax element	Meaning
CHF	Syntax initiator for a chamfer
1	Chamfer size as a fixed or variable number
F, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element

Example

```
7 L  X+0 Y+30  RL F300  M3
8 L  X+40 IY+5
9 CHF  12 F250
10 L  IX+5 Y+0
```

12.3.4 Rounding RND

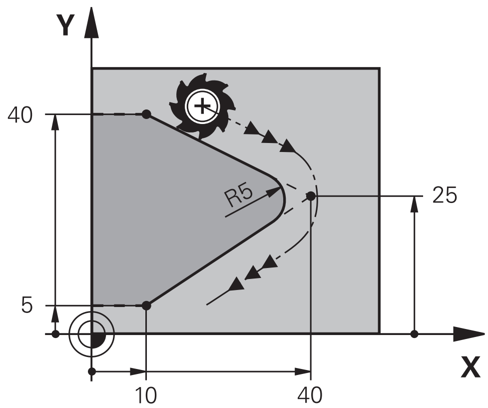
Application

The **RND** rounding arc function allows you to insert a rounding arc between two straight lines. The rounding arc is based on the intersection that you have programmed with the straight lines.

Requirements

- Path functions before and after the rounding arc
- Identical tool compensation before and after the rounding arc
- Rounding is machinable with the current tool

Description of function



You program the rounding arc between two path functions. The circular arc connects tangentially to the previous and subsequent contour element. The control does not traverse to the intersection.

If you program a feed rate in the **RND** block, then this feed rate is in effect only while cutting the rounding arc.

Input

11 RND R3 F200	; Radius with a size of 3 mm

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **RND**

The NC function includes the following syntax elements:

Syntax element	Meaning
RND	Syntax initiator for a radius
R	Radius size as a fixed or variable number
F, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element

Example

```
5 L  X+10 Y+40  RL F300  M3
6 L  X+40 Y+25
7 RND  R5 F100
8 L  X+10 Y+5
```

12.3.5 Circle center point CC

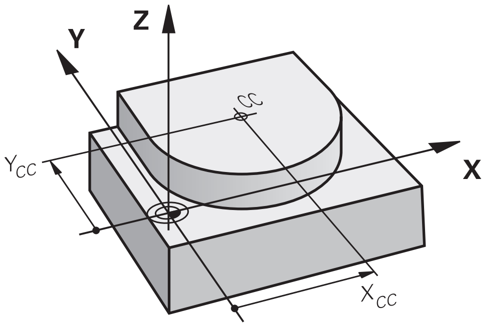
Application

The **CC** circle center function allows you to define a position as a circle center.

Related topics

- Programming a pole as a reference point for polar coordinates
  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



You define a circle center point by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The circle center point remains active until you define a new circle center point. The control does not traverse to the circle center point.

You need to define a circle center point before you can program a circular path with **C**.

> The control simultaneously uses the **CC** function as the pole for polar coordinates.
> **Further information:** "Polar coordinate datum at pole CC", Page 345

Input

11 CC X+0 Y+0	; Circle center

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **CC**

The NC function includes the following syntax elements:

Syntax element	Meaning
CC	Syntax initiator for a circle center
X, **Y**, **Z**, **U**, **V**, **W**	Coordinates of the circle center as a fixed or variable number Entry: absolute or incremental Optional syntax element

Example

```
5 CC  X+25 Y+25
```

or

```
10 L  X+25 Y+25
11 CC
```

12.3.6 Circular path C

Application

You use the circular path function **C** to program a circular path around a circle center point.

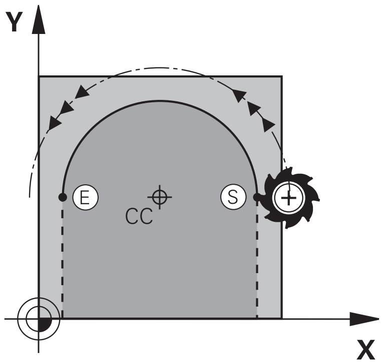
Related topics

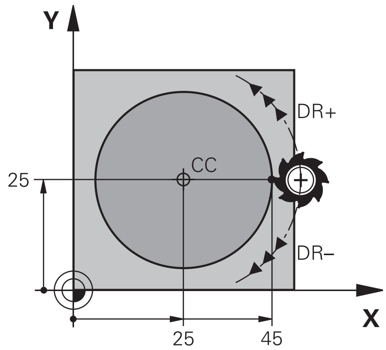
- Programming a circular path with polar coordinates
  Further information: "Circular path CP around pole CC", Page 348

Requirement

- Circle center point **CC** is defined
  Further information: "Circle center point CC", Page 332

Description of function





The control moves the tool on a circular path from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.

If you want to program a full circle, then define the same coordinates for the starting and end point. These points must lie on the circular path.

> In the machine parameter **circleDeviation** (no. 200901) you can define the permissible deviation of the circle radius. The maximum permissible deviation is 0.016 mm.

With the direction of rotation you define whether the control moves along the circular path in a clockwise or counterclockwise direction.

Definition of the direction of rotation:

- Clockwise: direction of rotation **DR-** (with radius compensation **RL**)
- Counterclockwise: direction of rotation **DR+** (with radius compensation **RL**)

Input

11 C X+50 Y+50 LIN_Z-3 DR- RL F250 M3	; Circular path with linear Z-axis superimpositioning

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **C**

The NC function includes the following syntax elements:

Syntax element	Meaning
C	Syntax initiator for a circular path around a circle center
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	End point of the circular path as a fixed or variable number Entry: absolute or incremental Optional syntax element
LIN_X, **LIN_Y**, **LIN_Z**, **LIN_A**, **LIN_B**, **LIN_C**, **LIN_U**, **LIN_V** or **LIN_W**	Axis and value of the linear superimpositioning as a fixed or variable number Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 341 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.

Further information: "Form column in the Program workspace", Page 227

Example

```
5 CC  X+25 Y+25
6 L  X+45 Y+25  RR F200  M3
7 C  X+45 Y+25  DR+
```

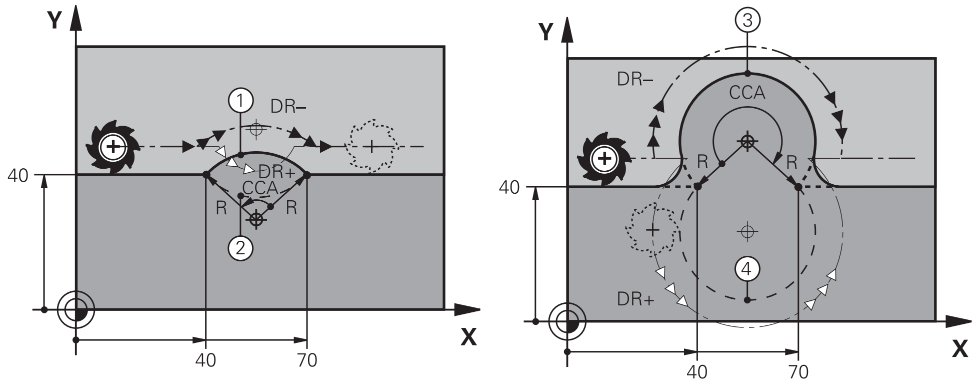
12.3.7 Circular path CR

Application

You use a radius to program a circular path with the circular path function **CR**.

Description of function

The control moves the tool on a circular path, with the radius **R**, from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.



The starting and end points can be connected with four different circular paths of the same radius. The correct circular path is defined with the **CCA** center angle of the circular path radius **R** and the direction of rotation **DR**.

The algebraic sign of the circular path radius **R** is decisive for whether the control selects a center angle that is greater than or less than 180°.

The radius has the following effects on the center angle:

- Smaller circular path: **CCA**<180°
  Radius with a positive sign **R**>0
- Longer circular path: **CCA**>180°
  Radius with a negative sign **R**<0

With the direction of rotation you define whether the control moves along the circular path in a clockwise or counterclockwise direction.

Definition of the direction of rotation:

- Clockwise: direction of rotation **DR-** (with radius compensation **RL**)
- Counterclockwise: direction of rotation **DR+** (with radius compensation **RL**)

10 L X+40 Y+40 RL F200 M3	
11 CR X+70 Y+40 R+20 DR-	; Circular path 1

or

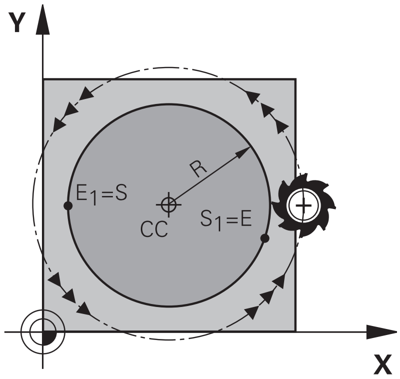
11 CR X+70 Y+40 R+20 DR+	; Circular path 2

or

11 CR X+70 Y+40 R-20 DR-	; Circular path 3

or

11 CR X+70 Y+40 R-20 DR+	; Circular path 4



For a full circle, program two circular paths in succession. The end point of the first circular path is the starting point of the second. The end point of the second circular path is the starting point of the first.

Input

11 CR X+50 Y+50 R+25 LIN_Z-2 DR- RL F250 M3	; Circular path with linear Z-axis superimpositioning

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **CR**

The NC function includes the following syntax elements:

Syntax element	Meaning
CR	Syntax initiator for a circular path with a radius
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	End point of the circular path as a fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius of the circular path as a fixed or variable number
LIN_X, **LIN_Y**, **LIN_Z**, **LIN_A**, **LIN_B**, **LIN_C**, **LIN_U**, **LIN_V** or **LIN_W**	Axis and value of the linear superimpositioning as a fixed or variable number Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 341 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

The distance between the starting and end points must not be greater than the circle diameter.

12.3.8 Circular path CT

Application

You use the circular path function **CT** to program a circular path that connects tangentially to the previously programmed contour element.

Related topics

- Programming a tangential connecting circular path with polar coordinates

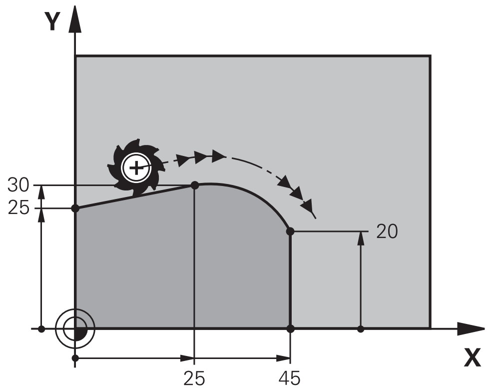
  Further information: "Circular path CTP", Page 350

Requirement

- Previous contour element programmed

  Before you can program a circular path with **CT** you must program a contour element to which the circular path can connect tangentially. This requires at least two NC blocks.

Description of function



The control moves the tool on a circular path, with a tangential connection, from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.

When contour elements uniformly merge into another without kinks, then this transition is referred to as tangential.

Input

11 CT X+50 Y+50 LIN_Z-2 RL F250 M3	; Circular path with linear Z-axis superimpositioning

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **CT**

The NC function includes the following syntax elements:

Syntax element	Meaning
CT	Syntax initiator for a circular path with a tangential connection
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	End point of the circular path as a fixed or variable number Entry: absolute or incremental Optional syntax element
LIN_X, **LIN_Y**, **LIN_Z**, **LIN_A**, **LIN_B**, **LIN_C**, **LIN_U**, **LIN_V** or **LIN_W**	Axis and value of the linear superimpositioning as a fixed or variable number Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 341 Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

- The contour element and the circular path should contain both coordinates of the plane in which the circular path is executed.
- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.

  Further information: "Form column in the Program workspace", Page 227

Example

```
7 L  X+0 Y+25  RL F300  M3
8 L  X+25 Y+30
9 CT  X+45 Y+20
10 L  Y+0
```

12.3.9 Linear superimpositioning of a circular path

Application

You can linearly superimpose a movement programmed in the working plane, thereby creating a spatial movement.

If, for example, you superimpose a circular path, you create a helix. A helix is a cylindrical spiral, such as a thread.

Related topics

- Linear superimpositioning of a circular path that is programmed with polar coordinates

  Further information: "Linear superimpositioning of a circular path", Page 352

Description of function

You can linearly superimpose the following circular paths:

- Circular contour **C**

  Further information: "Circular path C ", Page 334
- Circular contour **CR**

  Further information: "Circular path CR", Page 336
- Circular contour **CT**

  Further information: "Circular path CT", Page 339

> The tangential transition of the circular path **CT** has an effect only in the axes of the circular plane and not additionally on the linear superimpositioning.

In order to superimpose a linear movement onto circular paths with Cartesian coordinates, additionally program the optional syntax element **LIN**. You can define a main axis, rotary axis or parallel axis (e.g., **LIN_Z**).

Notes

- You can hide the input syntax element **LIN** in the settings in the **Program** workspace.

  Further information: "Settings in the Program workspace", Page 220
- Alternatively, you can also superimpose linear movements with a third axis, thereby creating a ramp. A ramp allows you, for example, to plunge into the material with a tool that is not a center-cut tool.

  Further information: "Straight line L", Page 326

Example

A program section repeat allows you to program a helix with the syntax element **LIN**.

This example shows an M8 thread with a depth of 10 mm.

The thread pitch is 1.25 mm. Thus, for a depth of 10 mm, eight thread grooves are required. An initial thread groove is also programmed as an approach path.

11 L Z+1.25 FMAX	; Pre-position in the tool axis
12 L X+4 Y+0 RR F500	; Pre-position in the plane
13 CC X+0 Y+0	; Activate the pole
14 LBL 1	
15 C X+4 Y+0 ILIN_Z-1.25 DR-	; Cut the first thread groove
16 LBL CALL 1 REP 8	; Mill the following eight thread grooves, **REP 8** = Number of remaining machining operations

This solution directly uses the thread pitch as the incremental infeed depth per revolution.

REP shows the number of repetitions required for reaching the calculated ten infeed runs.

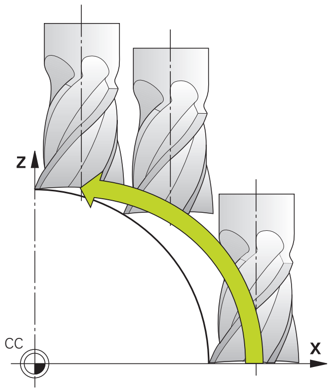
Further information: "Subprograms and program section repeats with the label LBL", Page 384

12.3.10 Circular path in another plane

Application

You can also program circular paths that do not lie in the active working plane.

Description of function



You program circular paths that lie in another plane by entering one axis of the working plane and the tool axis.

Further information: "Designation of the axes on milling machines", Page 208

You can program circular paths that lie in another plane with the following functions:

- **C**
- **CR**
- **CT**

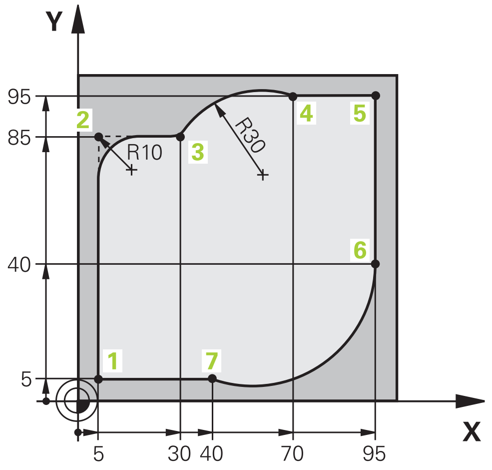
> If you want to use the function **C** for circular paths in another plane, you must first define the circle center point **CC** by entering one of the axes of the working plane and the tool axis.

Spatial arcs are created when these circular paths rotate. When machining spatial arcs, the control moves in three axes.

Example

```
3  TOOL CALL 1 Z S4000
4 ...
5 L  X+45 Y+25 Z+25 RR F200  M3
6 CC  X+25 Z+25
7 C  X+45 Z+25  DR+
```

12.3.11 Example: Cartesian path functions



0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Define the workpiece blank for workpiece simulation
3 TOOL CALL 1 Z S4000	; Call the tool in the tool axis and with the spindle speed
4 L Z+250 R0 FMAX	; Retract the tool in the tool axis at rapid traverse FMAX
5 L X-10 Y-10 R0 FMAX	; Pre-position the tool
6 L Z-5 R0 F1000 M3	; Move to working depth at feed rate F = 1000 mm/min
7 APPR LCT X+5 Y+5 R5 RL F300	; Approach the contour at point 1 on a circular path with tangential connection
8 L X+5 Y+85	; Program the first straight line for corner 2
9 RND R10 F150	; Program a rounding with R = 10 mm, feed rate F = 150 mm/min
10 L X+30 Y+85	; Move to point 3: starting point of the circular path CR
11 CR X+70 Y+95 R+30 DR-	; Move to point 4: end point of the circular path CR, with radius R = 30 mm
12 L X+95	; Move to point 5
13 L X+95 Y+40	; Move to point 6: starting point of the circular path CT
14 CT X+40 Y+5	; Move to point 7: end point of the circular path CT, arc with tangential connection to point 6; the control calculates the radius automatically
15 L X+5	; Move to last contour point 1
16 DEP LCT X-20 Y-20 R5 F1000	; Depart contour on a circular path with tangential connection
17 L Z+250 R0 FMAX M2	; Retract the tool, end program
18 END PGM CIRCULAR MM	

12.4 Path functions with polar coordinates

12.4.1 Overview of polar coordinates

With polar coordinates you can define a position in terms of its angle **PA** and its distance **PR** relative to a previously defined pole **CC**.

Overview of path functions with polar coordinates

Key	Function	Further information
L + P	Straight line **LP** (line polar)	Page 346
C + P	Circular path **CP** (circle polar) Circular path around circle center point or pole **CC** to arc end point	Page 348
CT + P	Circular path **CTP** (circle tangential polar) Circular path with tangential connection to the preceding contour element	Page 350
C + P	Helix with circular path **CP** (circle polar) Combination of a circular and a linear motion	Page 352

12.4.2 Polar coordinate datum at pole CC

Application

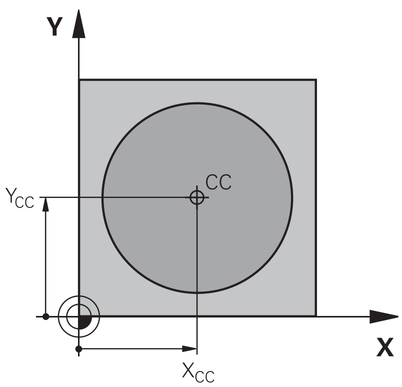
You must define a **CC** pole before programming with polar coordinates. All polar coordinates are relative to the pole.

Related topics

- Programming a circle center as a reference point for a circular path **C**

  Further information: "Circle center point CC", Page 332

Description of function



You use the **CC** function to define a position as the pole. You define a pole by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The pole remains active until you define a new pole. The control does not traverse to this position.

Input

11 CC X+0 Y+0	; Pole

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **CC**

The NC function includes the following syntax elements:

Syntax element	Meaning
CC	Syntax initiator for a pole
X, **Y**, **Z**, **U**, **V**, **W**	Coordinates of the pole as a fixed or variable number Entry: absolute or incremental Optional syntax element

Example

```
11 CC  X+30 Y+10
```

12.4.3 Straight line LP

Application

With the straight line function **LP** you program a straight traverse motion in any direction using polar coordinates.

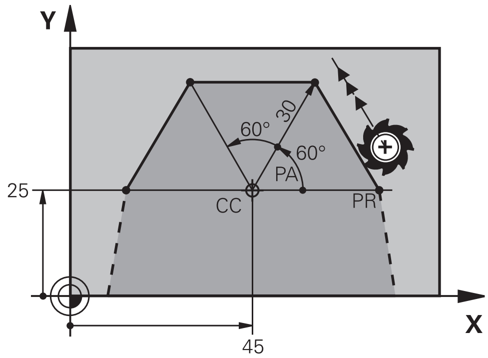
Related topics

- Programming a straight line with Cartesian coordinates
  Further information: "Straight line L", Page 326

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

You define the straight line with the polar coordinate radius **PR** and the polar coordinate angle **PA**. The polar coordinate radius **PR** is the distance from the end point to the pole.

The algebraic sign of **PA** depends on the angle reference axis:

- If the angle from the angle reference axis to **PR** is counterclockwise: **PA**>0
- If the angle from the angle reference axis to **PR** is clockwise: **PA**<0

Input

11 LP PR+50 PA+0 R0 FMAX M3	; Straight line without radius compensation in rapid traverse

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **L**

The NC function includes the following syntax elements:

Syntax element	Meaning
LP	Syntax initiator for a straight line with polar coordinates
PR	Polar coordinate radius as a fixed or variable number Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.

Further information: "Form column in the Program workspace", Page 227

Example

```
12 CC  X+45 Y+25
13 LP  PR+30 PA+0  RR F300  M3
14 LP  PA+60
15 LP  IPA+60
16 LP  PA+180
```

12.4.4 Circular path CP around pole CC

Application

You use the circular path function **CP** to program a circular path around the defined pole.

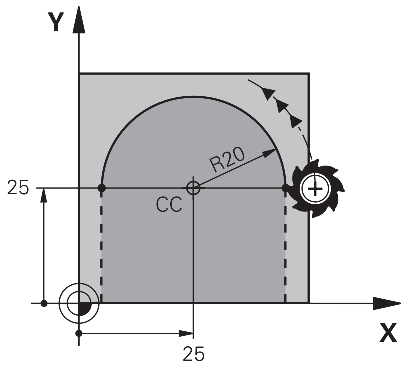
Related topics

- Programming a circular path with Cartesian coordinates
  Further information: "Circular path C ", Page 334

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



The control moves the tool on a circular path from the current position to the defined end point. The starting point is the end point of the preceding NC block.

The distance from the starting point to the pole is automatically both the polar coordinate radius **PR** as well as the radius of the circular path. You define the polar coordinate angle **PA** that the control moves to with this radius.

Input

11 CP PA+50 Z-2 DR- RL F250 M3	; Circular path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **C**

The NC function includes the following syntax elements:

Syntax element	Meaning
CP	Syntax initiator for a circular path around a pole
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Axis and value of the linear superimpositioning as a fixed or variable number Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 352 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Notes

- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.
- If you define **PA** incrementally, you must define the direction of rotation with the same algebraic sign.
  Consider this behavior when importing NC programs from earlier controls, and adapt the NC programs if necessary.

Example

```
18 LP  PR+20 PA+0  RR F250  M3
19 CC  X+25 Y+25
20 CP  PA+180 DR+
```

12.4.5 Circular path CTP

Application

You use the **CTP** function to program a circular path with polar coordinates that connects tangentially to the previously programmed contour element.

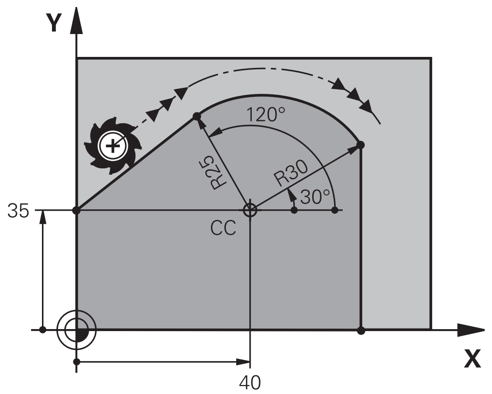
Related topics

- Programming a tangentially connecting circular path with Cartesian coordinates
  Further information: "Circular path CT", Page 339

Requirements

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 345
- Previous contour element programmed
  Before you can program a circular path with **CTP** you must program a contour element to which the circular path can connect tangentially. This requires at least two positioning blocks.

Description of function



The control moves the tool on a circular path, with a tangential connection, from the current position to the end point defined with polar coordinates. The starting point is the end point of the preceding NC block.

When contour elements uniformly merge into another, without kinks or corners, then this transition is referred to as tangential.

Input

11 CTP PR+30 PA+50 Z-2 DR- RL F250 M3	; Circular path

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ CT

The NC function includes the following syntax elements:

Syntax element	Meaning
CTP	Syntax initiator for a circular path with a tangential connection
PR	Polar coordinate radius as a fixed or variable number Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Axis and value of the linear superimpositioning as a fixed or variable number Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 352 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Notes

- The pole is **not** the center of the contour circle!
- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.

  Further information: "Form column in the Program workspace", Page 227

Example

```
12 L  X+0 Y+35  RL F250  M3
13 CC  X+40 Y+35
14 LP  PR+25 PA+120
15 CTP  PR+30 PA+30
16 L  Y+0
```

12.4.6 Linear superimpositioning of a circular path

Application

You can linearly superimpose a movement programmed in the working plane, thereby creating a spatial movement.

If, for example, you superimpose a circular path, you create a helix. A helix is a cylindrical spiral, such as a thread.

Related topics

- Linear superimpositioning of a circular path that is programmed with Cartesian coordinates

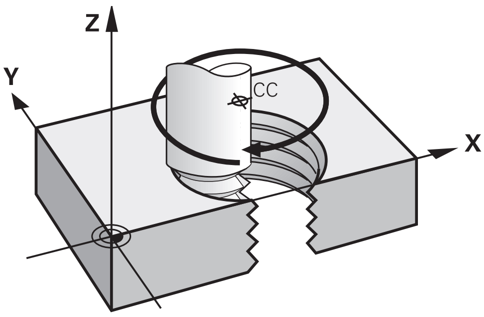
  Further information: "Linear superimpositioning of a circular path", Page 341

Requirements

The path contours for a helix can only be programmed with a circular path **CP**.

Further information: "Circular path CP around pole CC", Page 348

Description of function



A helix is a combination of a circular path **CP** and a linear motion perpendicular to this path. You program the circular path **CP** in the working plane.

Helices are used in the following cases:

- Large-diameter internal and external threads
- Lubrication grooves

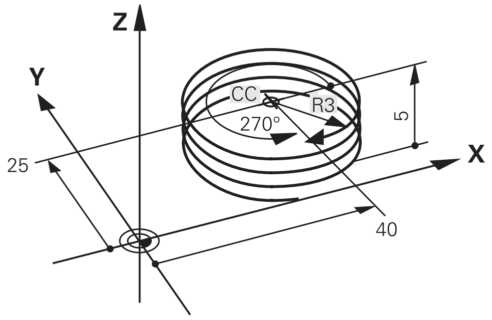
Dependencies of different thread shapes

The table shows the dependencies between machining direction, direction of rotation and radius compensation for the different thread shapes:

Internal thread	Work direction	Direction of rotation	Radius compensation
Right-handed	Z+	**DR+**	**RL**
	Z–	**DR-**	**RR**
Left-handed	Z+	**DR-**	**RR**
	Z–	**DR+**	**RL**

External thread	Work direction	Direction of rotation	Radius compensation
Right-handed	Z+	**DR+**	**RR**
	Z–	**DR-**	**RL**
Left-handed	Z+	**DR-**	**RL**
	Z–	**DR+**	**RR**

Programming a helix



> **i** Define the same algebraic sign for the direction of rotation **DR** and the incremental total angle **IPA**. The tool may otherwise move on a wrong path.

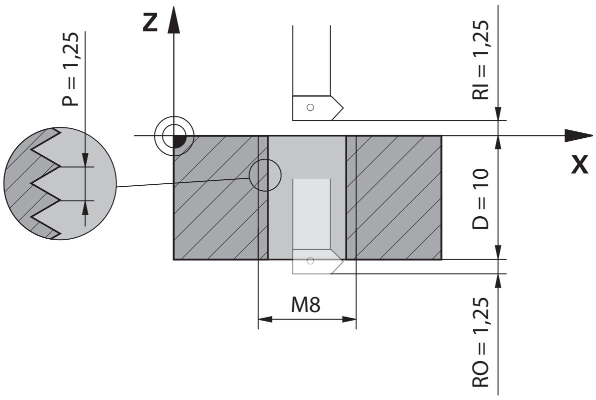
To program a helix:

- Select **C**

- Select **P**

- Select **I**
- Define the incremental total angle **IPA**
- Define the incremental total height **IZ**
- Select the direction of rotation
- Select radius compensation
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Example



This example includes the following default values:

- **M8** thread
- Left-handed thread miller

The drawing and the default values allow deriving the following information:

- Internal machining
- Right-hand thread
- **RR** radius compensation

The derived information requires the machining direction Z–.

Further information: "Dependencies of different thread shapes", Page 352

Specify and calculate the values below:

- Incremental total machining depth
- Number of thread grooves
- Incremental total angle

Formula	Definition
$IZ = D + RI + RO$	The incremental total machining depth **IZ** results from the thread depth **D** (depth) and from the optional thread run-in values **RI** (run-in) and thread run-out values **RO** (run-out).
$n = IZ \div P$	The number of thread grooves **n** (number) results from the incremental total machining depth **IZ** divided by the pitch **P** (pitch).
$IPA = n \times 360°$	The incremental total angle **IPA** results from the number of thread grooves **n** (number) multiplied by 360° for one complete revolution.

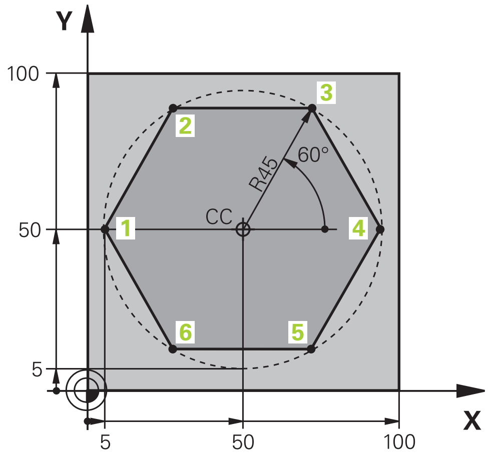
11 L Z+1,25 R0 FMAX	; Pre-position in the tool axis
12 L X+4 Y+0 RR F500	; Pre-position in the plane
13 CC X+0 Y+0	; Activate the pole
14 CP IPA-3600 IZ-12.5 DR-	; Cut the thread

Alternatively, you can also program the thread with a program section repeat.

Further information: "Subprograms and program section repeats with the label LBL", Page 384

Further information: "Example", Page 342

12.4.7 Example: Polar straight lines



0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Workpiece blank definition
3 TOOL CALL 1 Z S4000	; Tool call
4 CC X+50 Y+50	; Define the datum for polar coordinates
5 L Z+250 R0 FMAX	; Retract the tool
6 LP PR+60 PA+180 R0 FMAX	; Pre-position the tool
7 L Z-5 R0 F1000 M3	; Move to working depth
8 APPR PLCT PR+45 PA+180 R5 RL F250	; Approach the contour at point 1 on a circular path with tangential connection
9 LP PA+120	; Move to point 2
10 LP PA+60	; Move to point 3
11 LP PA+0	; Move to point 4
12 LP PA-60	; Move to point 5
13 LP PA-120	; Move to point 6
14 LP PA+180	; Move to point 1
15 DEP PLCT PR+60 PA+180 R5 F1000	; Depart contour on a circular path with tangential connection
16 L Z+250 R0 FMAX M2	; Retract the tool, end program
17 END PGM LINEARPO MM	

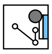
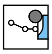
12.5 Fundamentals of approach and departure functions

Approach and departure functions allow you to avoid dwell marks on the workpiece because the tool gently approaches and departs from the contour.

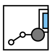
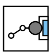
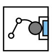
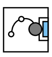
Because the approach and departure functions encompass multiple path functions, you get shorter NC programs. The defined syntax elements **APPR** and **DEP** make it easier for you to find contours in the NC program.

12.5.1 Overview of the approach and departure functions

The **APPR** folder of the **Insert NC function** window contains the following functions:

Symbol	Function	Further information
	**APPR LT** or **APPR PLT** Use Cartesian or polar coordinates to approach a contour on a straight line with a tangential connection	Page 358
	**APPR LN** or **APPR PLN** Use Cartesian or polar coordinates to approach a contour on a straight line perpendicular to the first contour point	Page 361
	**APPR CT** or **APPR PCT** Use Cartesian or polar coordinates to approach a contour on a circular path with a tangential connection	Page 363
	**APPR LCT** or **APPR PLCT** Use Cartesian or polar coordinates to approach a contour on a circular path with a tangential connection and a straight line	Page 365

The **DEP** folder of the **Insert NC function** window contains the following functions:

Symbol	Function	Further information
	**DEP LT** Depart contour on a straight line with a tangential connection	Page 367
	**DEP LN** Depart contour on a straight line perpendicular to the last contour point	Page 368
	**DEP CT** Depart contour on a circular path with a tangential connection	Page 369
	**DEP LCT** or **DEP PLCT** Use Cartesian or polar coordinates to depart a contour on a circular path with a tangential connection and a straight line	Page 369

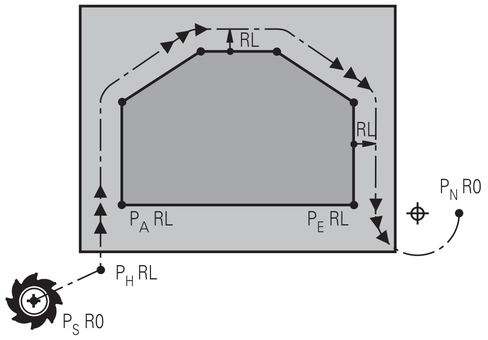
> You can switch between entry of Cartesian and polar coordinates in the form or by pressing the **P** key.
>
> **Further information:** "Fundamentals of coordinate definitions", Page 318

Approaching or departing a helix

The tool approaches and departs a helix in the extension of the helix by moving on a circular path that connects tangentially to the contour. Use the **APPR CT** and **DEP CT** functions for this.

Further information: "Linear superimpositioning of a circular path", Page 352

12.5.2 Positions for approach and departure



NOTICE

Danger of collision!

The control traverses from the current position (starting point P_S) to the auxiliary point P_H at the last feed rate entered. If you programmed **FMAX** in the last positioning block before the approach function, the control also approaches the auxiliary point P_H at rapid traverse.

- Program a feed rate other than **FMAX** before the approach function

The control uses the following positions when approaching and departing a contour:

- Starting point P_S

  The starting point P_S is programmed prior to the approach function without radius compensation. The starting point is located outside of the contour.

- Auxiliary point P_H

  Certain approach and departure functions require an additional auxiliary point P_H. The control automatically calculates the auxiliary point using the entered information.

  In order to determine the auxiliary point P_H, the control requires a subsequent path function. If no path function follows, then the control stops the machining operation or simulation with an error message.

- First contour point P_A

  Program the first contour point P_A within the approach function, along with the radius compensation **RR** or **RL**.

  > If you program **R0**, then the control may stop the machining operation or simulation with an error message.
  > This reaction is different from the behavior of the iTNC 530.

- Last contour point P_E

  You program the last contour point P_E with any path function.

- End point P_N

  The position P_N is located outside of the contour and arises from the information entered within the departure function. The departure function automatically cancels the radius compensation.

NOTICE

Danger of collision!

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

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

Definitions

Abbreviation	Definition
APPR (approach)	Approach function
DEP (departure)	Departure function
L (line)	Line segment
C (circle)	Circle
T (tangential)	Continuous, smooth transition
N (normal)	Perpendicular line

12.6 Approach and departure functions with Cartesian coordinates

12.6.1 Approach function APPR LT

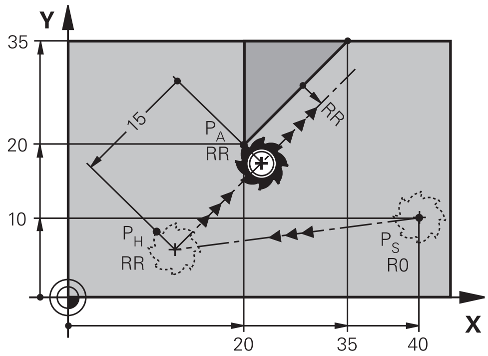
Application

With the **APPR LT** NC function, the control approaches the contour on a straight line tangential to the first contour element.
Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PLT** with polar coordinates
  Further information: "Approach function APPR PLT", Page 372

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H
- A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR LT X+20 Y+20 LEN15 RR F300	; Approach the contour in a tangential linear path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR LT**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR LT	Syntax initiator for a linear approach function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR LT

11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	; Approach P_A with **RR**, distance P_H to P_A: **LEN15**
13 L X+35 Y+35	; Complete the first contour element

12.6.2 Approach function APPR LN

Application

With NC function **APPR LN**, the control approaches the contour on a straight line perpendicular to the first contour element.

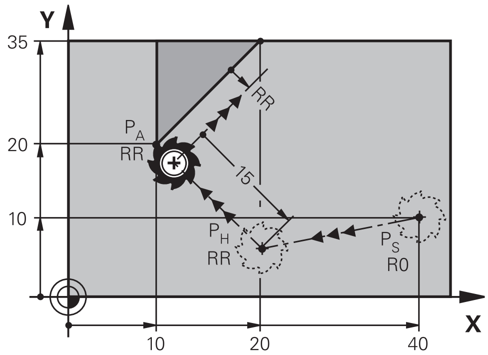
Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PLN** with polar coordinates

  Further information: "Approach function APPR PLN", Page 374

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H
- A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR LN X+20 Y+20 LEN+15 RR F300	; Linearly and perpendicularly approach the contour

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR LN**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR LN	Syntax initiator for a linear approach function perpendicular to the contour
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR LN

11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR LN X+10 Y+20 Z-10 LEN+15 RR F100	; Approach P_A with **RR**; distance: P_H to P_A: **LEN+15**
13 L X+20 Y+35	; Complete the first contour element

12.6.3 Approach function APPR CT

Application

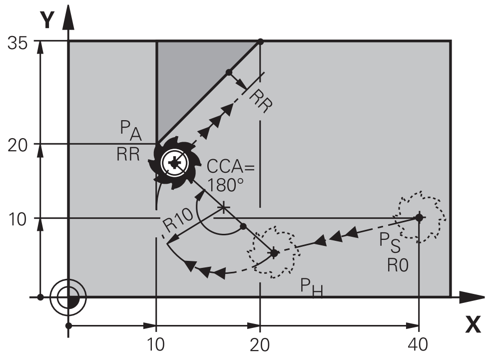
With NC function **APPR CT**, the control approaches the contour on a circular path tangential to the first contour element.

Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PCT** with polar coordinates
  Further information: "Approach function APPR PCT", Page 376

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H
  The distance of the auxiliary point P_H to the first contour point P_A arises from the center angle **CCA** and the radius **R**.
- A circular path from the auxiliary point P_H to the first contour point P_A
  The circular path is defined by the center angle **CCA** and the radius **R**.
  The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 APPR CT X+20 Y+20 CCA80 R+5 RR F300	; Approach the contour in a tangential circular path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR CT**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR CT	Syntax initiator for a circular approach function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
CCA	Center angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR CT

11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR CT X+10 Y+20 Z-10 CCA180 R +10 RR F100	; Approach P_A with **CCA180** and **RR**; distance P_H to P_A: **R+10**
13 L X+20 Y+35	; Complete the first contour element

12.6.4 Approach function APPR LCT

Application

With the NC function **APPR LCT**, the control approaches the contour on a straight line, followed by a circular path tangential to the first contour element.

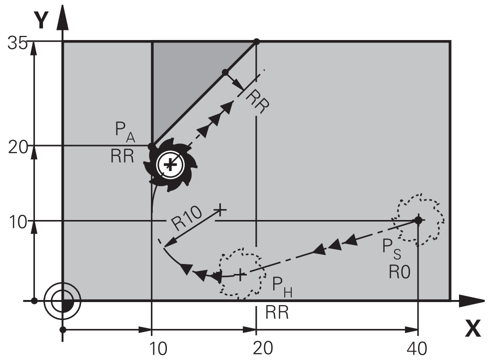
Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PLCT** with polar coordinates

  Further information: "Approach function APPR PLCT", Page 379

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H

  The straight line is tangential to the circular path.

  The auxiliary point P_H is determined based on the starting point P_S, the radius **R** and the first contour point P_A.
- A circular path in the working plane from the auxiliary point P_H to the first contour point P_A

  The circular path is uniquely defined by the radius **R**.

If you program the Z coordinates in the approach function, then the tool approaches simultaneously in three axes from the starting point P_S to the auxiliary point P_H.

Input

11 APPR LCT X+20 Y+20 Z-10 R5 RR F300	; Approach the contour in a tangential circular path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR LCT**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR LCT	Syntax initiator for a linear and circular approach function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR LCT

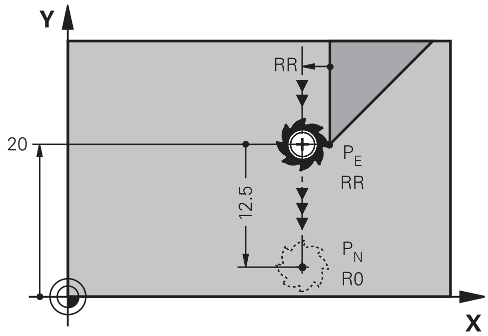
11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR LCT X+10 Y+20 Z-10 R10 RR F100	; Approach P_A with **RR**; distance P_H to P_A: **R10**
13 L X+20 Y+35	; Complete the first contour element

12.6.5 Departure function DEP LT

Application

With NC function **DEP LT**, the control departs from the contour on a straight line tangential to the last contour element.

Description of function



The tool moves in a straight line from the last contour point P_E to the end point P_N.

Input

11 DEP LT LEN5 F300	; Depart from the contour in a tangential linear path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP LT**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP LT	Syntax initiator for a linear departure function tangential to the contour
LEN	Distance of the auxiliary point P_H to the contour Fixed or variable number Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Example DEP LT

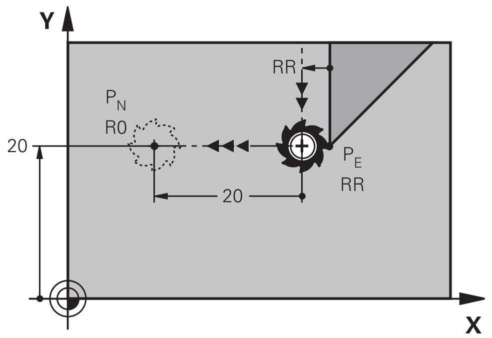
11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LT LEN12.5 F100	; Approach P_N; distance P_E to P_N: **LEN12.5**

12.6.6 Departure function DEP LN

Application

With the NC function **DEP LN**, the control departs from the contour in a straight line perpendicular to the last contour element.

Description of function



The tool moves in a straight line from the last contour point P_E to the end point P_N.

The distance from the end point P_N to the contour point P_E is **LEN** plus the tool radius.

Input

11 DEP LN LEN+10 F300	; Depart from the contour in a perpendicular linear path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP LN**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP LN	Syntax initiator for a linear departure function perpendicular to the contour
LEN	Distance of the auxiliary point P_H to the contour Fixed or variable number Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Example DEP LN

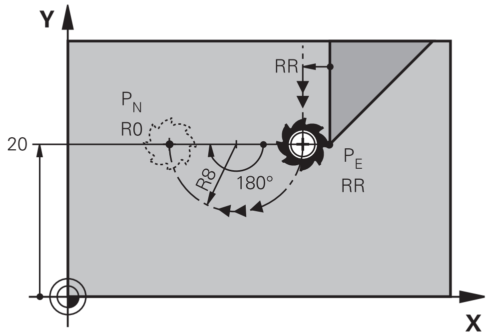
11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LN LEN+20 F100	; Approach P_N; distance P_E to P_N: **LEN+20**

12.6.7 Departure function DEP CT

Application

With the NC function **DEP CT**, the control departs from the contour in a circular path tangential to the last contour element.

Description of function



The tool moves on a circular path from the last contour point P_E to the end point P_N.

The circular path is defined by the center angle **CCA** and the radius **R**.

The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 DEP CT CCA30 R+8	; Depart from the contour on a tangential circular path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP CT**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP CT	Syntax initiator for a circular departure function tangential to the contour
CCA	Center angle as a fixed or variable number
R	Radius as a fixed or variable number
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Example DEP CT

11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP CT CCA180 R+8 F100	; Approach P_N with **CCA180**; distance P_E to P_N: **R+8**

12.6.8 Departure function DEP LCT

Application

With the NC function **DEP LCT**, the control departs from the contour on a circular path, followed by a tangential straight line to the last contour element.

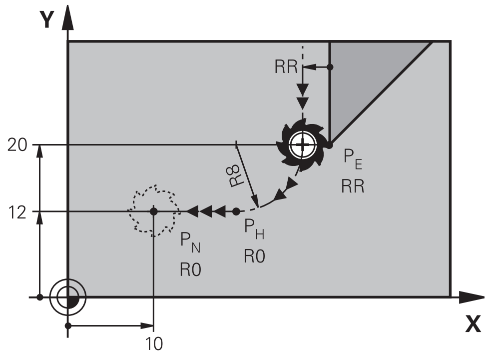
The coordinates of the end point P_N are programmed with Cartesian coordinates.

Related topics

- **DEP LCT** with polar coordinates

  Further information: "Departure function DEP PLCT", Page 380

Description of function



This NC function encompasses the following steps:

- On a circular path from the last contour point P_E to the auxiliary point P_H

  The auxiliary point P_H is determined based on the last contour point P_E, the radius **R** and the end point P_N.
- On a straight line from the auxiliary point P_H to the end point P_N

If you program the Z coordinate in the departure function, then the tool moves simultaneously in three axes from the auxiliary point P_H to the end point P_N.

Input

11 DEP LCT X-10 Y-0 R15	; Tangentially depart from the contour linearly and circularly

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP LCT**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP LCT	Syntax initiator for a linear and circular departure function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Coordinates of the last contour point Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example DEP LCT

11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LCT X+10 Y+12 R8 F100	; Approach P_N; distance P_E to P_N: **R8**

12.7 Approach and departure functions with polar coordinates

12.7.1 Approach function APPR PLT

Application

With the **APPR PLT** NC function, the control approaches the contour on a straight line tangential to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

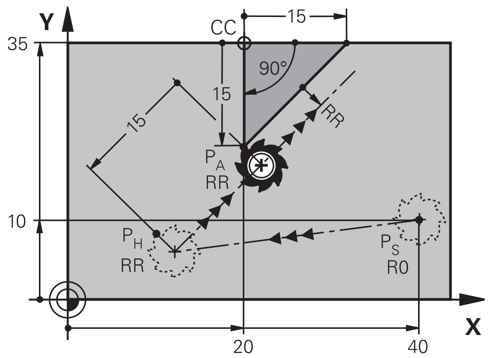
Related topics

- **APPR LT** with Cartesian coordinates
  Further information: "Approach function APPR LT", Page 358

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H
- A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR PLT PR+15 PA-90 LEN15 RR F200	; Approach the contour in a tangential linear path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR PLT**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PLT	Syntax initiator for a linear approach function tangential to the contour
PR	Polar coordinate radius as a fixed or variable number Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR PLT

11 L X+10 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PLT PR+30 PA+180 LEN10 RL F300	; Approach P_A with **RL**; distance from P_H to P_A: **LEN10**
14 LP PR+30 PA+125	; Complete the first contour element

12.7.2 Approach function APPR PLN

Application

With NC function **APPR PLN**, the control approaches the contour on a straight line perpendicular to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

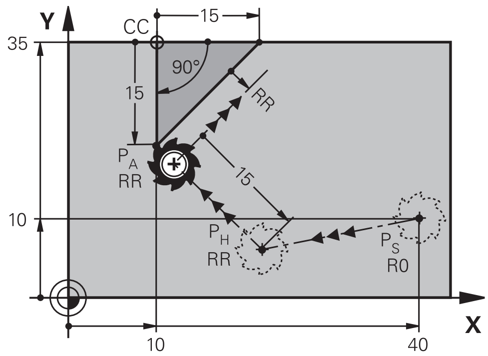
Related topics

- **APPR LN** with Cartesian coordinates
  Further information: "Approach function APPR LN", Page 361

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H
- A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR PLN PR+15 PA-90 LEN+15 RL F300	; Linearly and perpendicularly approach the contour

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR PLN**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PLN	Syntax initiator for a linear approach function perpendicular to the contour
PR	Polar coordinate radius as a fixed or variable number Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Entry: absolute or incremental Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR PLN

11 L X-5 Y+25 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PLN PR+30 PA+180 LEN+10 RL F300	; Approach P_A with **RL**; P_H to P_A; **LEN+10**
14 LP PR+30 PA+125	; Complete the first contour element

12.7.3 Approach function APPR PCT

Application

With NC function **APPR PCT**, the control approaches the contour on a circular path tangential to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

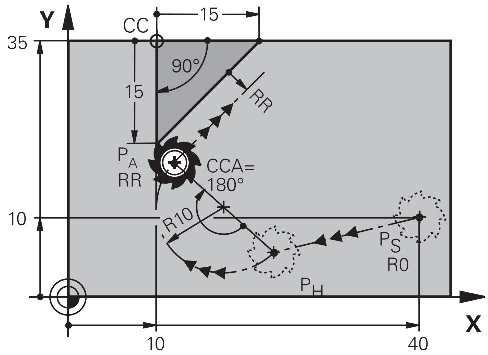
Related topics

- **APPR CT** with Cartesian coordinates
  Further information: "Approach function APPR CT", Page 363

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H

  The distance of the auxiliary point P_H to the first contour point P_A arises from the center angle **CCA** and the radius **R**.
- A circular path from the auxiliary point P_H to the first contour point P_A

  The circular path is defined by the center angle **CCA** and the radius **R**.

  The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

> If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 APPR PCT PR+15 PA-90 CCA180 R +10 RL F300	; Approach the contour in a tangential circular path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR PCT**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PCT	Syntax initiator for a circular approach function tangential to the contour
PR	Polar coordinate radius as a fixed or variable number Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
CCA	Center angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR PCT

11 L X+5 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PCT PR+30 PA+180 CCA40 R +20 RL F300	; Approach P_A with **CCA40** and **RL**; distance P_H to P_A: **R+20**
14 LP PR+30 PA+125	; Complete the first contour element

12.7.4 Approach function APPR PLCT

Application

With the NC function **APPR PLCT**, the control approaches the contour in a straight line, followed by a circular path tangential to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

Related topics

- **APPR LCT** with Cartesian coordinates

  Further information: "Approach function APPR LCT", Page 365

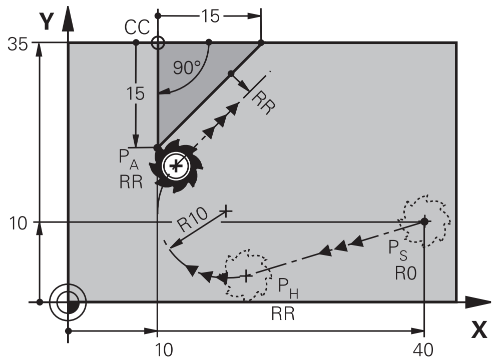
Requirement

- Pole **CC**

  You must define a pole **CC** before programming with polar coordinates.

  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H

  The straight line is tangential to the circular path.

  The auxiliary point P_H is determined based on the starting point P_S, the radius **R** and the first contour point P_A.

- A circular path in the working plane from the auxiliary point P_H to the first contour point P_A

  The circular path is uniquely defined by the radius **R**.

If you program the Z coordinates in the approach function, then the tool approaches simultaneously in three axes from the starting point P_S to the auxiliary point P_H.

Input

11 APPR PLCT PR+15 PA-90 R10 RL F300	; Tangentially approach the contour linearly and circularly

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR PLCT**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PLCT	Syntax initiator for a linear and circular approach function tangential to the contour
PR	Polar coordinate radius as a fixed or variable number Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example APPR PLCT

11 L X+10 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PLCT PR+30 PA+180 R20 RL F300	; Approach P_A with **RL**; P_H to P_A: **R20**
14 LP PR+30 PA+125	; Complete the first contour element

12.7.5 Departure function DEP PLCT

Application

With the NC function **DEP PLCT**, the control departs from the contour on a circular path, followed by a tangential straight line to the last contour element.

The coordinates of the end point P_N are programmed with polar coordinates.

Related topics

- **DEP LCT** with Cartesian coordinates

  Further information: "Departure function DEP LCT", Page 370

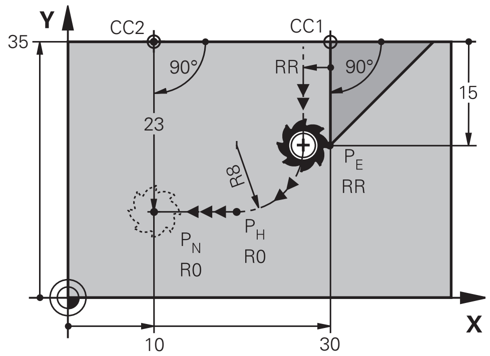
Requirement

- Pole **CC**

  You must define a pole **CC** before programming with polar coordinates.

  Further information: "Polar coordinate datum at pole CC", Page 345

Description of function



This NC function encompasses the following steps:

- On a circular path from the last contour point P_E to the auxiliary point P_H

  The auxiliary point P_H is determined based on the last contour point P_E, the radius **R** and the end point P_N.
- On a straight line from the auxiliary point P_H to the end point P_N

If you program the Z coordinate in the departure function, then the tool moves simultaneously in three axes from the auxiliary point P_H to the end point P_N.

Input

11 DEP PLCT PR15 PA-90 R8	; Tangentially depart from the contour linearly and circularly

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP PLCT**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP PLCT	Syntax initiator for a linear and circular departure function tangential to the contour
PR	Polar coordinate radius as a fixed or variable number Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate as a fixed or variable number **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Miscellaneous function as a fixed or variable number **Further information:** "Miscellaneous Functions", Page 1317 Optional syntax element

Note

You can switch between the syntax for Cartesian and polar coordinate input in the **Form** column.

Further information: "Form column in the Program workspace", Page 227

Example DEP PLCT

11 CC X+50 Y+20	; Set the pole
12 LP PR+30 PA+0 RL F300	; Approach the last contour element P_E with **RL**
13 DEP PLCT PR+50 PA+0 R5	; Approach P_N; distance P_E to P_N: **R5**

13

Programming Techniques

13.1 Subprograms and program section repeats with the label LBL

Application

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as necessary. Use subprograms to insert contours or complete machining steps after the end of the program and call them in the NC program. Program section repeats repeat single or several NC blocks during the NC program. Subprograms and program section repeats can also be combined.

Subprograms and program section repeats are programmed with the NC function **LBL**.

Related topics

- Executing NC programs within another NC program
  Further information: "Calling an NC program with PGM CALL", Page 388
- Jumps with conditions as if-then decisions.
  Further information: " Jump commands folder", Page 1380

Description of function

The label **LBL** is used for defining the machining steps for subprograms and program section repeats.

The control offers the following keys and icons in connection with labels:

Key or icon	Function
LBL SET	Create **LBL**
LBL CALL	Call **LBL**: Jump to the label in the NC program
	In case of **LBL** number: Enter the next free number automatically

Defining a label with LBL SET

The **LBL SET** function defines a new label in the NC program.

Each label must be unambiguously identifiable in the NC program by a number or a name. If a number or a name exists twice in an NC program, the control shows a warning ahead of the NC block.

LBL 0 marks the end of a subprogram. This number is the only one which may exist any number of times in the NC program.

Input

11 LBL "Reset"	; Subprogram for resetting a coordinate transformation
12 TRANS DATUM RESET	
13 LBL 0	

The NC function includes the following syntax elements:

Syntax element	Meaning
LBL	Syntax initiator for a label
0 or **" "**	Number or name of the label. Fixed or variable number or name Input: **0...65535** or **text width 32** Use an icon to enter the next free number automatically. **Further information:** "Description of function", Page 384

Calling a label with CALL LBL

The **CALL LBL** function calls a label in the NC program.

When the control reads **CALL LBL**, it jumps to the defined label and continues executing the NC program from this NC block. When the control reads **LBL 0**, it jumps back to the next NC block after **CALL LBL**.

In case of program section repeats, you can optionally define that the control executes that jump several times.

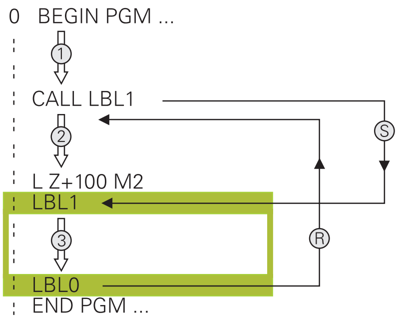
Input

11 CALL LBL 1 REP2	; Call label 1 twice

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL LBL	Syntax initiator for calling a label
Number, **" "** or **QS**	Number or name of the label. Fixed or variable number or name Input: **1...65535** or **text width 32** or **0...1999** The label can be selected from a selection menu including all labels available in the NC program.
REP	Number of repetitions until the control executes the next NC block Optional syntax element

Subprograms



A subprogram allows calling parts of an NC program any number of times at different points of the NC program (e.g., machining positions or a contour).

A subprogram starts with a **LBL** label and ends with **LBL 0**. **CALL LBL** calls the subprogram from any point in the NC program. In this process, repetitions must not be defined with **REP**.

The control executes the NC program as follows:

1 The control executes the NC program up to the **CALL LBL** function.
2 The control jumps to the beginning of the defined subprogram **LBL**.
3 The control executes the subprogram up to the subprogram end **LBL 0**.
4 After that, the control jumps to the next NC block after **CALL LBL** and continues executing the NC program.

The following conditions apply to subprograms:

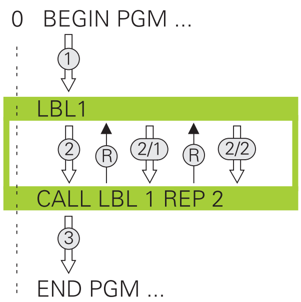
- A subprogram cannot call itself
- **CALL LBL 0** is not permitted (Label 0 is only used to mark the end of a subprogram).
- Write subprograms after the NC block with M2 or M30

  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

The control displays information about the active subprogram on the **LBL** tab of the **Status** workspace.

Further information: "LBL tab", Page 175

Program-section repeats



A program section repeat allows repeating a part of an NC program any number of times (e.g., contour machining with incremental infeed).

A program section repeat starts with a **LBL** label and ends after the last programmed repetition **REP** of the label call **CALL LBL**.

The control executes the NC program as follows:

1 The control executes the NC program up to the **CALL LBL** function.

   In this process, the control already executes the program section once because the program section to be repeated is positioned ahead of the **CALL LBL** function.

2 The control jumps to the beginning of the program section repeat **LBL**.

3 The control repeats the program section as many times as programmed under **REP**.

4 After that, the control continues executing the NC program.

The following conditions apply to program section repeats:

- Program the program section repeat before the end of the program with **M30** or **M2**.
- No **LBL 0** can be defined with a program section repeat.
- 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.

The control displays information about the active program section repeat on the **LBL** tab of the **Status** workspace.

Further information: "LBL tab", Page 175

Notes

- The control displays the NC function **LBL SET** in the structure by default.

  Further information: "Structure column in the Program workspace", Page 1514
- You can repeat a program section up to 65 534 times in succession
- The following characters are allowed in the name of a label: # $ % & , - _ . 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
- The following characters are not allowed in the name of a label: <blank> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~
- Before creating your NC program, compare the subprogram and program section repeat programming techniques using if-then decisions.

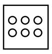
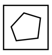
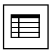
  You can thereby avoid possible misunderstandings and programming errors.

  Further information: " Jump commands folder", Page 1380

13.2 Selection functions

13.2.1 Overview of selection functions

The **Selection** folder of the **Insert NC function** window contains the following functions:

Icon	Function	Further information
PGM CALL	Call an NC program with **PGM CALL**	Page 388
	Select a datum table with **SEL TABLE**	Page 1034
	Select a point table with **SEL PATTERN**	Page 401
	Select a contour program with **SEL CONTOUR**	Page 413
	Select an NC program with **SEL PGM**	Page 390
	Call the last selected file with **CALL SELECTED PGM**	Page 390
CYC	Select any NC program with **SEL CYCLE** as a machining cycle	Page 481
	Select a correction table with **SEL CORR-TABLE**	Page 1120
	Open the file with **OPEN FILE**	Page 1159
	Link multiple contours with **CONTOUR DEF**	Page 406

13.2.2 Calling an NC program with PGM CALL

Application

The **PGM CALL** function lets you call another separate NC program from an already existing NC program. The control executes the called NC program at the point where you called it in the NC program. This allows a machining operation to be executed with various transformations, for example.

Related topics

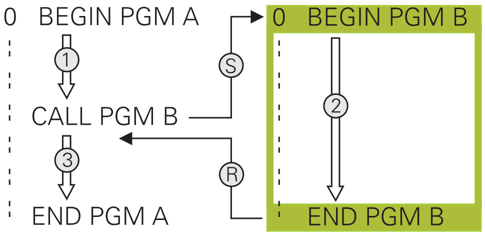
- Program call with Cycle **12 PGM CALL**

  Further information: "Cycle 12 PGM CALL ", Page 395
- Program call following selection

  Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 390
- Executing several NC programs as a job list

  Further information: "Pallet Machining and Job Lists", Page 1937

Description of function



The control executes the NC program as follows:

1. The control executes the calling NC program until you call another NC program with **CALL PGM**.
2. After that, the control executes the called NC program up to the last NC block.
3. The control then resumes the calling NC program, starting with the next NC block after **CALL PGM**.

The following conditions apply to program calls:

- The called NC program must not contain a **CALL PGM** call into the calling NC program. This creates an endless loop.
- The called NC program must not contain a miscellaneous function **M30** or **M2**. If you have defined subprograms in the called NC program using labels, then you can replace **M30** or **M2** with an unconditional jump function. This keeps the control from executing a subprogram.

  Further information: "Unconditional jump", Page 1381

  If the called NC program contains the miscellaneous functions, the control generates an error message.
- The called NC program must be complete. If the NC block **END PGM** is missing, the control outputs an error message.

Input

11 CALL PGM reset.h	; Call NC program

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL PGM	Syntax initiator for calling an NC program
reset.h	Path of the called NC program The NC program can be selected in a selection menu.

Notes

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 program call path including the name of the NC program may contain no more than 255 characters.
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- If you want to program variable program calls in connection with string parameters, use the **SEL PGM** function.
- If you wish to program variable program calls in conjunction with string parameters, then use the function **SEL PGM**.

  Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 390
- With a **PGM CALL** program call, Q parameters always have a global effect. So please note that changes to Q parameters in the called NC program can also influence the calling NC program. If applicable, use QL parameters that take effect only in the active NC program.
- As a rule, Q parameters are active globally with a **PGM CALL**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program. If required, use QL parameters which affect only the active NC program.
- While the control is executing the calling NC program, editing of all called NC programs is disabled.

13.2.3 Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM

Application

The function **SEL PGM** allows selecting another separate NC program that you can call at a different position in the active NC program. The control executes the selected NC program at the position where you call it in the calling NC program using **CALL SELECTED PGM**.

Related topics

- Calling the NC program directly

  Further information: "Calling an NC program with PGM CALL", Page 388

Description of function

The control executes the NC program as follows:

1 The control executes the NC program until another NC program is called with **CALL PGM**. When the control reads **SEL PGM**, it remembers the defined NC program.
2 When the control reads **CALL SELECTED PGM**, it calls the NC program previously selected at this point.
3 After that, the control executes the called NC program up to the last NC block.
4 Then the control continues executing the calling NC program with the next NC block after **CALL SELECTED PGM**.

The following conditions apply to program calls:

- The called NC program must not contain a **CALL PGM** call into the calling NC program. This creates an endless loop.
- The called NC program must not contain a miscellaneous function **M30** or **M2**. If you have defined subprograms in the called NC program using labels, then you can replace **M30** or **M2** with an unconditional jump function. This keeps the control from executing a subprogram.

  Further information: "Unconditional jump", Page 1381

  If the called NC program contains the miscellaneous functions, the control generates an error message.
- The called NC program must be complete. If the NC block **END PGM** is missing, the control outputs an error message.

Input

11 SEL PGM "reset.h"	; Select an NC program for calling
*** - ...**	
21 CALL SELECTED PGM	; Call the selected NC program

The NC function **SEL PGM** includes the following syntax elements:

Syntax element	Meaning
SEL PGM	Syntax initiator for selecting an NC program to be called
" " or **QS**	Path of the called NC program Fixed or variable name The NC program can be selected in a selection menu.

The NC function **CALL SELECTED PGM** includes the following syntax elements:

Syntax element	Meaning
CALL SELECTED PGM	Syntax for calling the selected NC program

Notes

- Within the **SEL PGM** function, the NC program can also be selected with QS parameters so that the program call can be variably controlled.
- If an NC program called by **CALL SELECTED PGM** is missing, the control interrupts the execution or simulation of the program with an error message. In order to avoid undesired interruptions during program run, you can use the function **FN 18: SYSREAD** (**ID10 NR110** and **NR111**) to check all paths at the beginning of the program.

  Further information: "Read system data with FN 18: SYSREAD", Page 1388
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- As a rule, Q parameters are active globally with a **PGM CALL**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program. If required, use QL parameters which affect only the active NC program.
- While the control is executing the calling NC program, editing of all called NC programs is disabled.

13.3 NC sequences for reuse

Application

You can save up to 200 successive NC blocks as NC sequences and insert them during programming using the **Insert NC function** window. In contrast to the called NC programs, you can adapt the NC sequences after insertion without changing the actual sequence.

Related topics

- The **Insert NC function** window

  Further information: "Inserting NC functions", Page 228
- Selecting and copying the NC blocks with the context menu

  Further information: "Context menu", Page 1522
- Calling the NC programs unchanged

  Further information: "Calling an NC program with PGM CALL", Page 388

Description of function

You can use NC blocks in the **Editor** operating mode and the **MDI** application.

The control saves the NC sequences as complete NC programs in the **TNC:\system\PGM-Templates** folder. You can also create subfolders in order to sort the NC sequences.

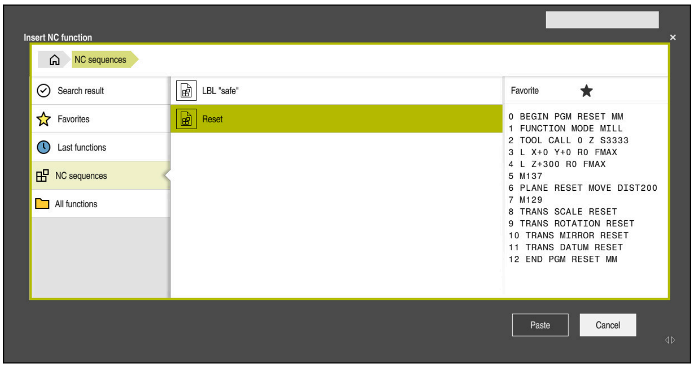
Here are the following possibilities for creating an NC sequence:

- Save marked NC blocks with the **Create NC sequence** button

  Further information: "Context menu in the Program workspace", Page 1525
- Create a new NC program in the **TNC:\system\PGM-Templates** folder
- Copy the already existing NC program to the **TNC:\system\PGM-Templates** folder

If you create an NC sequence with the **Create NC sequence** button, then the control opens the **Save NC sequence** window. In this window, you define the name of the NC sequence.

The control displays all NC sequences alphabetically in the **Insert NC function** window under **NC sequences**. You can insert the desired NC sequence at the cursor position and in the NC program.



NC sequences in the **Insert NC function** window

If you open an NC sequence as its own tab in the **Editor**, then you can permanently change the content of the NC sequence.

Notes

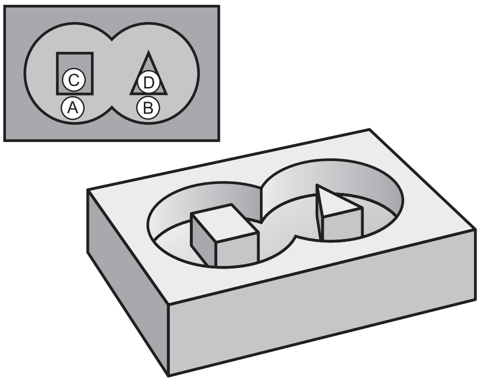
- For every NC sequence, you must define a unique name. If you try to save an NC sequence under a name that has already been assigned, then the control opens the **Overwrite NC sequence** window. The control asks if you wish to overwrite the existing NC sequence.
- If you select an NC sequence in the **Insert NC function** window and swipe to the right, then the control offers the following file functions:
  - Edit
  - Rename
  - Delete
  - Open the path in the **Files** operating mode
  - Mark as favorite
- If you create a backup of the **TNC:** partition with the **NC/PLC Backup** function, then the backup also contains the NC sequences.

  Further information: "Backup and restore", Page 2148

13.4 Cycle 14 CONTOUR

ISO programming
G37

Application



In Cycle **14 CONTOUR**, list all subprograms that are to be superimposed to define the overall contour .

Related topics

- Simple contour formula
  Further information: "Simple contour formula", Page 406
- Complex contour formula
  Further information: "Complex contour formula", Page 410
- Overlapping contours
  Further information: "Superimposed contours", Page 402

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **14** is DEF-active which means that it becomes effective as soon as it is defined in the NC program.
- You can list up to 12 subprograms (subcontours) in Cycle **14**.

13.4.1 Cycle parameters

Help graphic	Parameter
	Label numbers for contour? Enter all label numbers for the individual subprograms that are to be superimposed to define a contour. Confirm each number with the ENT key. Confirm your entries with the **END** key. Up to 12 subprogram numbers are possible. Input: **0...65535**

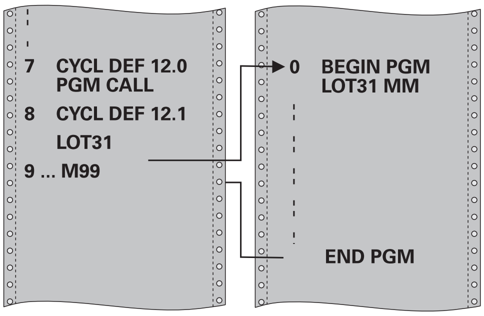
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL1 /2
```

13.5 Cycle 12 PGM CALL

ISO programming
G39

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: "Selection functions", Page 388

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- 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**.
- If you want to define an ISO program to be a cycle, add the .I file type to the program name.

13.5.1 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 File Select in the action bar of the NC program to be called.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare NC program 1_Plate.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\OCM\1_Plate.h
13 L  X+20  Y+50 R0 FMAX M99
```

13.6 Nesting of programming techniques

Application

Programming techniques can also be combined with one another. For example, another separate NC program or a subprogram can be called in a program section repeat.

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

Related topics

- Subprograms
  Further information: "Subprograms", Page 386
- Program section repeats
  Further information: "Program-section repeats", Page 387
- Calling a separate NC program
  Further information: "Selection functions", Page 388

Description of function

The following maximum nesting depths apply to NC programs:

- 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

13.6.1 Example

Subprogram call within a subprogram

0 BEGIN PGM UPGMS MM	
*** - ...**	
11 CALL LBL "UP1"	; Call subprogram **LBL "UP1"**
*** - ...**	
21 L Z+100 R0 FMAX M30	; Last program block of main program with M30
22 LBL "UP1"	; Start of subprogram **"UP1"**
*** - ...**	
31 CALL LBL 2	; Call subprogram **LBL 2**
*** - ...**	
41 LBL 0	; End of sub program **"UP1"**
42 LBL 2	; Start of subprogram **LBL 2**
*** - ...**	
51 LBL 0	; End of subprogram **LBL 2**
52 END PGM UPGMS MM	

The control executes the NC program as follows:

1 NC program UPGMS is executed up to NC block 11.
2 Subprogram UP1 is called and executed up to NC block 31.
3 Subprogram 2 is called, and executed up to NC block 51. 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 32 up to NC block 41. End of subprogram UP1 and return jump to NC program UPGMS.
5 NC program UPGMS is executed from NC block 12 up to NC block 21. Program end with return jump NC block 1.

Program-section repeat within a program section repeat

0 BEGIN PGM REPS MM	
* - ...	
11 LBL 1	; Start of program section 1
* - ...	
21 LBL 2	; Start of program section 2
* - ...	
31 CALL LBL 2 REP 2	; Call program section 2 and repeat twice
* - ...	
41 CALL LBL 1 REP 1	; Call program section 1 including program section 2 and repeat once
* - ...	
51 END PGM REPS MM	

The control executes the NC program as follows:

1 NC program REPS is executed up to NC block 31.
2 The program section between NC block 31 and NC block 21 is repeated twice, meaning that it is executed three times in total.
3 NC program REPS is executed from NC block 32 up to NC block 41.
4 The program section between NC block 41 and NC block 11 is repeated once, meaning that it is executed twice in total (including the program section repeat between NC block 21 and NC block 31).
5 NC program REPS is executed from NC block 42 up to NC block 51. Program end with return jump to NC block 1.

Subprogram call within a program section repeat

0 BEGIN PGM UPGREP MM	
* - ...	
11 LBL 1	; Start of program section 1
12 CALL LBL 2	; Call subprogram 2
13 CALL LBL 1 REP 2	; Call program section 1 and repeat twice
* - ...	
21 L Z+100 R0 FMAX M30	; Last NC block of main program with M30
22 LBL 2	; Start of subprogram 2
* - ...	
31 LBL 0	; End of subprogram 2
32 END PGM UPGREP MM	

The control executes the NC program as follows:

1 NC program UPGREP is executed up to NC block 12.
2 Subprogram 2 is called, and executed up to NC block 31.
3 The program section between NC block 13 and NC block 11 (including subprogram 2) is repeated twice, meaning that it is executed three times in total.
4 NC program UPGREP is executed from NC block 14 up to NC block 21. Program end with return jump to NC block 1.

14

Contour and Point Definitions

14.1 Point tables

Application

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

Related topics

- Contents of a point table, hiding individual points

  Further information: "Point table", Page 2044

Description of function

Coordinates in a point table

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting point coordinates of the respective cycle, e.g. center coordinates of a circular pocket. The coordinates of the spindle axis correspond to the coordinate of the workpiece surface.

The control retracts the tool to the clearance height when traversing between the starting points. Depending on which is greater the control uses either the tool axis coordinate from the cycle call or the value from cycle parameter **Q204 2ND SET-UP CLEARANCE**.

NOTICE

Danger of collision!

If you program a clearance height for individual points in a point table, the control will ignore the value from the cycle parameter **Q204 2ND SET-UP CLEARANCE** for all points!

▶ Program the function **GLOBAL DEF 125 POSITIONING** so that the control will take into account the clearance height only for the respective point.

Effect with cycles

SL cycles and Cycle 12

The control interprets the points in the point table as an additional datum shift.

Cycles 200 to 208, 262 to 267

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 tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

Cycles 210 to 215

The control interprets the points as an additional datum shift. If you want to use the points defined in the point table as the starting point coordinates, you must program the starting points and the coordinate of the workpiece upper edge (**Q203**) in the respective milling cycle as 0.

> You can no longer insert these cycles on the control, but you can edit and run them in existing NC programs.

Cycles 251 to 254

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 tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

14.1.1 Selecting the point table in the NC program with SEL PATTERN

To select the point table:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **SEL PATTERN**
- Select **File selection**
- > The control opens a window for the file selection.
- Select the desired point table through the file structure
- Confirm your input
- > The control concludes the NC block.

If the point table is not stored in the same directory as the NC program, you must define the complete path name. In the **Program settings** window you can define whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 220

Example

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

14.1.2 Calling the cycle with a point table

If you want to call a cycle at the points that you defined in the point table, then program the cycle call with **CYCLE CALL PAT**.

CYCL CALL PAT enables the control to execute the point table that you defined last.

To call a cycle in conjunction with a point table:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.

CYCL CALL

- Select **CYCL CALL PAT**
- Enter a feed rate

> The control will use this feed rate to traverse between the points of the point table. If you do not enter a feed rate, the control moves the tool at the feed rate last defined.

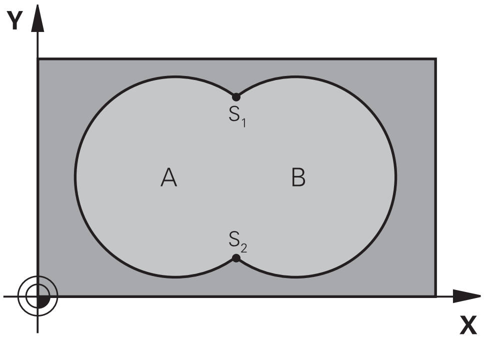
- Define miscellaneous functions, if necessary
- Confirm your input with the **END** key

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**.

14.2 Superimposed contours

14.2.1 Fundamentals



Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Related topics

- Cycle 14 **CONTOUR**
  Further information: "Cycle 14 CONTOUR ", Page 394

14.2.2 Subprograms: overlapping pockets

> The following examples show contour subprograms that are called by Cycle **14 CONTOUR** in a main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2. They need not be programmed.

The pockets are programmed as full circles.

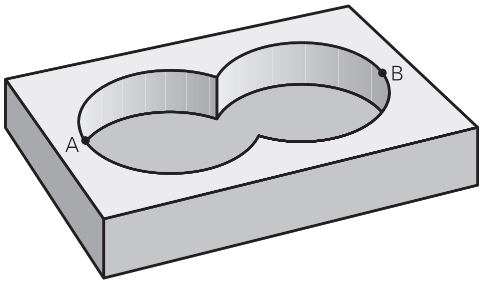
Subprogram 1: Pocket A

```
11 LBL 1
12 L  X+10  Y+10 RR
13 CC  X+35  Y+50
14 C  X+10  Y+50 DR-
15 LBL 0
```

Subprogram 2: Pocket B

16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0

14.2.3 Surface resulting from sum



Both surfaces A and B are to be machined, including the overlapping area:

- The surfaces A and B must be pockets
- The first pocket (in Cycle **14**) must start outside the second pocket

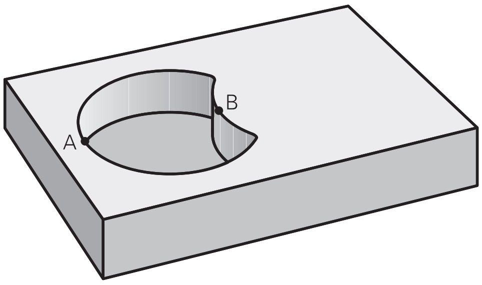
Surface A:

11 LBL 1
12 L X+10 Y+50 RR
13 CC X+35 Y+50
14 C X+10 Y+50 DR-
15 LBL 0

Surface B:

16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0

14.2.4 Surface resulting from difference



Surface A is to be machined without the portion overlapped by B:

- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.

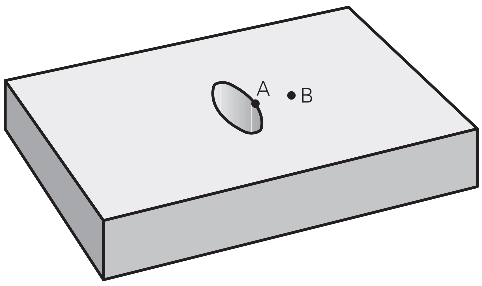
Surface A:

```
11 LBL 1
12 L  X+10  Y+50 RR
13 CC  X+35  Y+50
14 C  X+10  Y+50 DR-
15 LBL 0
```

Surface B:

```
16 LBL 2
17 L  X+40  Y+50 RL
18 CC  X+65  Y+50
19 C  X+40  Y+50 DR-
20 LBL 0
```

14.2.5 Surface resulting from intersection



Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- A and B must be pockets
- A must start inside of B

Surface A:

```
11 LBL 1
12 L  X+60  Y+50 RR
13 CC  X+35  Y+50
14 C  X+60  Y+50 DR-
15 LBL 0
```

Surface B:

```
16 LBL 2
17 L  X+90  Y+50 RR
18 CC  X+65  Y+50
19 C  X+90  Y+50 DR-
20 LBL 0
```

14.3 Simple contour formula

14.3.1 Fundamentals

Program structure: Machining with SL Cycles and simple contour formula

```
0 BEGIN CONTDEF MM
...
5 CONTOUR DEF
...
6 CYCL DEF 20 CONTOUR DATA
...
8 CYCL DEF 21 ROUGH-OUT
...
9 CYCL CALL
...
13 CYCL DEF 23 FLOOR FINISHING
...
14 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING
...
17 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 END PGM CONTDEF MM
```

Using simple contour formulas, you can easily combine up to nine subcontours (pockets or islands) to program a particular contour. The control calculates the complete contour from the selected subcontours.

> The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.

Void areas

Using optional void areas **V** (**void**), you can exclude areas from machining. These area can be, for example, contours in castings or areas machined in previous steps. You can define up to five void areas.

If you are using OCM cycles, the control will plunge vertically within void areas.

If you are using SL Cycles **22** to **24**, the control will determine the plunging position, regardless of any defined void areas.

Run the simulation to verify proper behavior.

Properties of the subcontours

- Do not program radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are permitted; if they are programmed within the subcontours, they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored..
- The working plane is defined in the first coordinate block of the subprogram.

Cycle properties

- The control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions; the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed; the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations).
- The contour is approached on a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

The machining dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **20 CONTOUR DATA** or **271 OCM CONTOUR DATA**.

14.3.2 Entering a simple contour formula

You can use the selection possibility in the action bar or in the form to interlink various contours in a mathematical formula.

Proceed as follows:

Insert NC function

- Select **Insert NC function**
- The control opens the **Insert NC function** window.
- Select **CONTOUR DEF**
- The control opens the dialog for entering the contour formula.
- Enter the first subcontour **P1**
- Select the **P2** pocket or **I2** island selection possibility
- Enter second subcontour
- If needed, enter the depth of the second subcontour.
- Carry on with the dialog as described above until you have entered all subcontours.
- Define void areas **V** as needed

> The depth of the void areas corresponds to the total depth that you define in the machining cycle.

You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a QS parameter
LBL	■ Number ■ Name ■ QS	Define the number, name or QS parameter for a label

Example:

```
11 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 DEPTH5 V1 = LBL 3
```

> Programming notes:
> - The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth Therefore, always start programming the subcontour with the deepest pocket.
> - If the contour is defined as an island, the control interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!
> - If you enter a value of 0 for the depth, then the depth defined in Cycle **20** is effective for pockets. For islands, this means that they extend up to the workpiece surface!
> - 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.

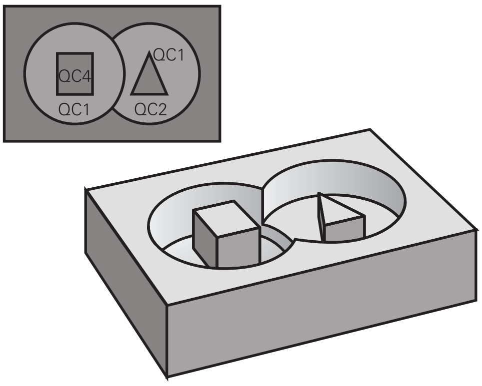
14.3.3 Machining contours with SL or OCM cycles

The defined entire contour is machined with the SL cycles or the OCM cycles (see "Overview", Page 507).

14.4 Complex contour formula

14.4.1 Fundamentals

Using complex contour formulas, you can combine several subcontours (pockets or islands) to program complex contours. You define the individual subcontours (geometry data) in separate NC programs. In this way, any subcontour can be reused any number of times. The control calculates the complete contour from the selected subcontours, which you link through a contour formula.



Program structure: Machining with SL Cycles and complex contour formula

0 BEGIN CONT MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 CONTOUR DATA
...
8 CYCL DEF 21 ROUGH-OUT
...
9 CYCL CALL
...
13 CYCL DEF 23 FLOOR FINISHING
...
14 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING
...
17 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 END PGM CONT MM

Programming notes:

- The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.
- To use SL cycles with contour formulas, it is mandatory that your program is structured carefully. These cycles enable you to save frequently used contours in individual NC programs. Using the contour formula, you can connect the subcontours to define a complete contour and specify whether it applies to a pocket or island.

Properties of the subcontours

- The control assumes that each contour is a pocket. Thus, do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are permitted—if they are programmed within the subcontours, they are also effective in the NC programs called subsequently. However, they need not be reset after the cycle call.
- Although the called NC programs can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the NC program.
- Subcontours can be defined with different depths according to your requirements.

Cycle properties

- The control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions; the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling

The machining dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **20 CONTOUR DATA** or **271 OCM CONTOUR DATA**.

Program structure: Calculation of the subcontours with contour formula

```
0 BEGIN MODEL MM
1 DECLARE CONTOUR QC1 = "120"
2 DECLARE CONTOUR QC2 = "121" DEPTH15
3 DECLARE CONTOUR QC3 = "122" DEPTH10
4 DECLARE CONTOUR QC4 = "123" DEPTH5
5 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
6 END PGM MODEL MM
```

```
0 BEGIN PGM 120 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM 120 MM
```

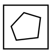
```
0 BEGIN PGM 121 MM
...
```

14.4.2 Selecting an NC program with contour definition

With the **SEL CONTOUR** function, you select an NC program with contour definitions, from which the control extracts the contour descriptions:

Proceed as follows:

- Select **Insert NC function**
- The control opens the **Insert NC function** window.



- Select **SEL CONTOUR**
- The control opens the dialog for entering the contour formula.
- Definition of the contour

You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a string parameter

Programming notes:

- 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.
- Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR** is no longer necessary if you use **SEL CONTOUR**.

14.4.3 Defining a contour description

Using the **DECLARE CONTOUR** function in your NC program, you enter the path for NC programs from which the control extracts the contour descriptions. In addition, you can select a separate depth for this contour description.

Proceed as follows:

Insert NC function

- Select **Insert NC function**
- The control opens the **Insert NC function** window.
- Select **DECLARE CONTOUR**
- The control opens the dialog for entering the contour formula.
- Enter the number for the contour designator **QC**
- Defining a contour description

You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a string parameter

Programming notes:

- With the entered contour designators **QC** you can include the various contours in the contour formula.
- 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.
- If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).
- The control will only take different depths (**DEPTH**) into account if the elements overlap. In case of pure islands inside a pocket, this is not the case. Use a simple contour formula for this purpose.

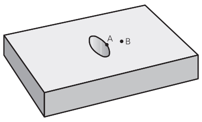
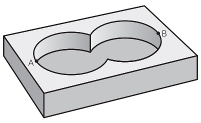
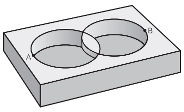
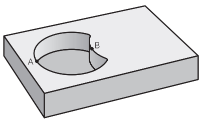
  Further information: "Simple contour formula", Page 406

14.4.4 Entering a complex contour formula

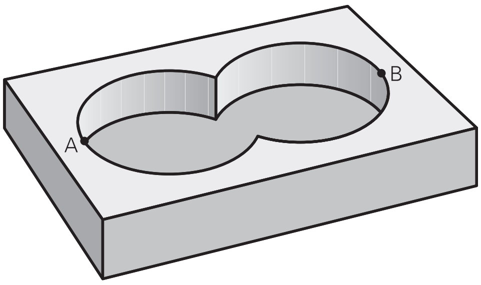
You can use the contour formula function to interlink various contours in a mathematical formula.

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **Contour formula QC**
- > The control opens the dialog for entering the contour formula.
- Enter the number for the contour designator **QC**
- Entering a contour formula

Help graphic	Input	Mathematical function	Example
	**&**	Intersected with	**QC10 = QC1 & QC5**
	**|**	Joined with	**QC25 = QC7 | QC18**
	**^**	Joined with, but w/o intersection	**QC12 = QC5 ^ QC25**
	****	Without	**QC25 = QC1 \\ QC2**
	(	Opening parenthesis	**QC12 = QC1 & (QC2 | QC3)**
	)	Closing parenthesis	**QC12 = QC1 & (QC2 | QC3)**
		Defining a single contour	**QC12 = QC1**

14.4.5 Superimposed contours



By default, the control considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: overlapping pockets

> The following examples are contour description programs that are defined in a contour definition program. The contour definition program is called through the **SEL CONTOUR** function in the actual main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

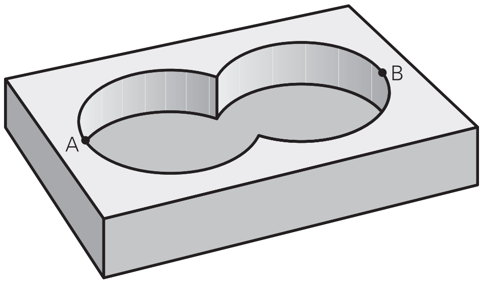
Contour description program 1: pocket A

```
0  BEGIN PGM POCKET MM
1  L  X+10  Y+50 R0
2  CC  X+35  Y+50
3  C  X+10  Y+50 DR-
4  END PGM POCKET MM
```

Contour description program 2: pocket B

```
0  BEGIN PGM POCKET2 MM
1  L  X+90  Y+50 R0
2  CC  X+65  Y+50
3  C  X+90  Y+50 DR-
4  END PGM POCKET2 MM
```

Area of inclusion



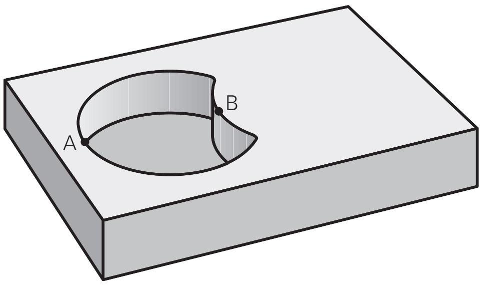
Both areas A and B are to be machined, including the overlapping area:

- Areas A and B must have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "joined with" function.

Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 | QC2
* - ...
```

Area of exclusion



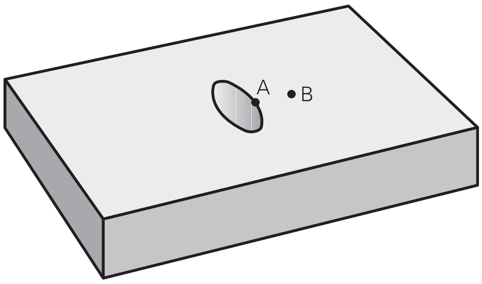
Area A is to be machined without the portion overlapped by B:

- Surfaces A and B must be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the area B is subtracted from the area A with the **without** function.

Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 \ QC2
* - ...
```

Area of intersection



Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- Surfaces A and B must be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "intersection with" function.

Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 & QC2
* - ...
```

14.4.6 Machining contours with SL or OCM cycles

> The defined entire contour is machined with the SL cycles or the OCM cycles (see "Overview", Page 507).

14.5 Pattern definition with PATTERN DEF

14.5.1 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**

Possible setting	Definition	Further information
POS1	Point Definition of up to any 9 machining positions	Page 421
ROW1	Row Definition of a single row, straight or rotated	Page 422
PAT1	Pattern Definition of a single pattern, straight, rotated or distorted	Page 423
FRAME1	Frame Definition of a single frame, straight, rotated or distorted	Page 425
CIRC1	Circle Definition of a full circle	Page 427
PITCH-CIRC1	Pitch circle Definition of a pitch circle	Page 428

14.5.2 Entering PATTERN DEF

Proceed as follows:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **PATTERN DEF**
- > The control opens the dialog for entering **PATTERN DEF**.
- Select the desired machining pattern, e.g. **CIRC1** for a full circle
- Enter the required definitions
- Define machining cycle, e.g. cycle **200 DRILLING**
- Call cycle with **CYCL CALL PAT**

14.5.3 Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function.

Further information: "Programming a machining cycle", Page 145

The control performs the most recently defined machining cycle on the machining pattern you defined.

Program structure: Machining with PATTERN DEF

```
0 BEGIN SL 2 MM
...
11 PATTERN DEF POS1 (X+25 Y+33.5 Z+0) POS2 (X+15 IY+6.5 Z+0)
12 CYCL DEF 200 DRILLING
...
13 CYCL CALL PAT
```

Notes

Programming note

- Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with **Q345=1**. Then, between the holes, the control always positions the tool to the 2nd set-up clearance that was defined in the cycle.

Operating notes:

- A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.

  Further information: "Selecting the point table in the NC program with SEL PATTERN", Page 401
- 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**.
- You can use the mid-program startup function to select any point at which you want to start or continue machining.

  Further information: "Block scan for mid-program startup", Page 1965

14.5.4 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 )
```

14.5.5 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 )
```

Related topics

- Cycle **221 CARTESIAN PATTERN** (ISO **G221**)
  Further information: "Cycle 221 CARTESIAN PATTERN ", Page 435

14.5.6 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 )
```

Related topics

- Cycle **221 CARTESIAN PATTERN** (ISO **G221**)
  Further information: "Cycle 221 CARTESIAN PATTERN ", Page 435

14.5.7 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 )
```

14.5.8 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 )
```

Related topics

- Cycle **220 POLAR PATTERN** (ISO **G220**)
  Further information: "Cycle 220 POLAR PATTERN ", Page 432

14.5.9 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 the selection possibility on the action bar or in the form) 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 )
```

Related topics
- Cycle **220 POLAR PATTERN** (ISO **G220**)
  Further information: "Cycle 220 POLAR PATTERN ", Page 432

14.5.10 Example: Using cycles in connection 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: "Technology-independent cycles", Page 488 and "Cycles for milling"

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, end program
20 M30	
21 END PGM 1 MM	

14.6 Cycles for pattern definition

14.6.1 Overview

The control provides three cycles for machining point patterns:

Cycle		Call	Further information
220	**POLAR PATTERN** ■ Defining a circular pattern ■ Full circle or pitch circle ■ Input of start and end angles	**DEF**-active	Page 432
221	**CARTESIAN PATTERN** ■ Defining a linear pattern ■ Input of an angle of rotation	**DEF**-active	Page 435
224	**DATAMATRIX CODE PATTERN** ■ Converting text to a DataMatrix code to be used as a point pattern ■ Input of position and size	**DEF**-active	Page 439

14.6.2 Cycle 220 POLAR PATTERN

ISO programming
G220

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 427
- Defining a circle segment with **PATTERN DEF**
  Further information: "Defining a pitch circle", Page 428

Cycle sequence

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 or a circular arc. 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 **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

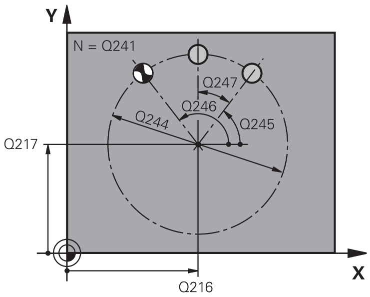
Notes

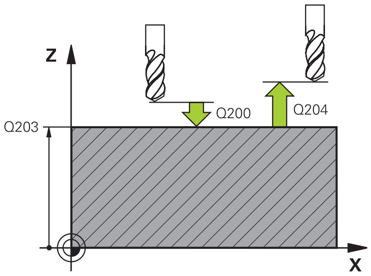
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **220** is DEF-active. In addition, Cycle **220** automatically calls the last defined machining cycle.

Note on programming

- If you combine one of the machining cycles **200** to **209** or **251** to **267** 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** or **PREDEF**
	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** or **PREDEF**
	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**
	Q365 Type of traverse? Line=0/arc=1 Specify how the tool moves between machining processes: **0**: Move between operations on a straight line **1**: Move between operations on the pitch circle Input: **0**, **1**

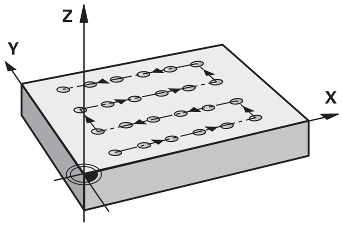
Example

```
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=+0           ;SURFACE COORDINATE ~
    Q204=+50          ;2ND SET-UP CLEARANCE ~
    Q301=+1           ;MOVE TO CLEARANCE ~
    Q365=+0           ;TYPE OF TRAVERSE
12 CYCL CALL
```

14.6.3 Cycle 221 CARTESIAN PATTERN

ISO programming
G221

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 422
- Defining an individual pattern with **PATTERN DEF**
  Further information: "Defining an individual pattern", Page 423

Cycle sequence

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 **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

Notes

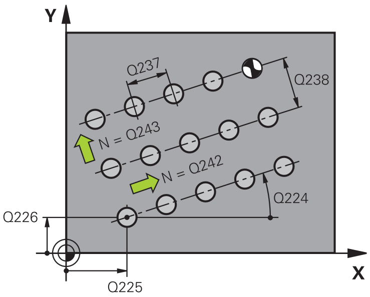
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined machining cycle.

Notes on programming

- If you combine Cycle **221** with one of the machining cycles **200** to **209** or **251** to **267**, 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.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.

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?

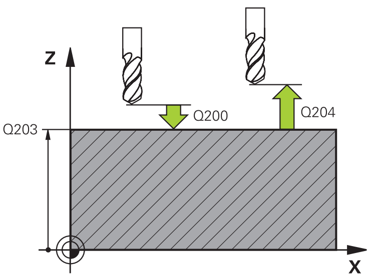
Number of lines

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** or **PREDEF**

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** or **PREDEF**

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
```

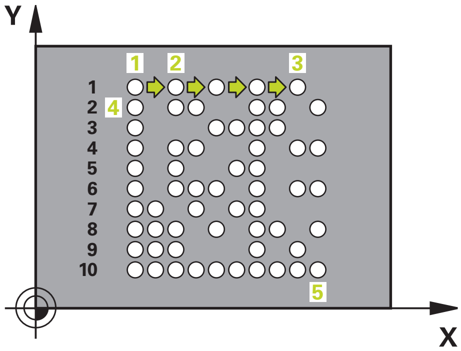
14.6.4 Cycle 224 DATAMATRIX CODE PATTERN

ISO programming
G224

Application

Use Cycle **224 DATAMATRIX CODE PATTERN** to convert text to a so-called DataMatrix code. This code will be used as a point pattern for a previously defined fixed cycle.

Cycle sequence



1 The control automatically moves the tool from its current position to the programmed starting point. This point is always located in the lower left corner.
  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 Then, the control moves the tool in the positive direction of the secondary axis to the first point **1** in the first row
3 From this position, the control executes the last defined fixed machining cycle
4 Then, the control moves the tool in the positive direction of the principal axis to point **2** for the next operation.
5 This procedure will be repeated until all machining operations in the first row have been completed. The tool is located above the last point **3** of the first row
6 Then, the control moves the tool in the negative direction of the principal and secondary axes to the first point **4** of the next row
7 Then, the next points are machined
8 These steps are repeated until the entire DataMatrix code has been completed. Machining stops in the lower right corner **5**
9 Finally, the control retracts the tool to the programmed 2nd set-up clearance

Notes

NOTICE

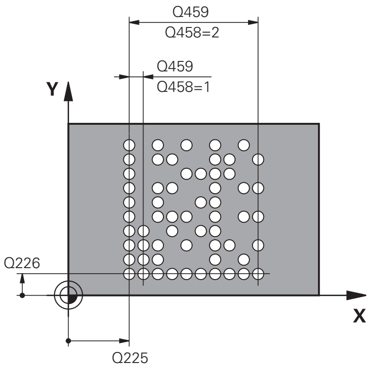
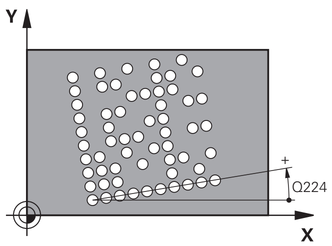
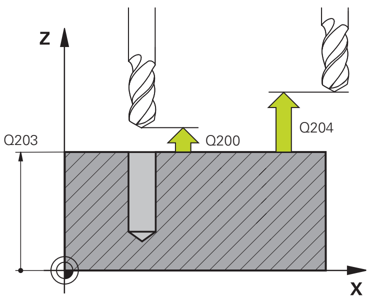
Danger of collision!

If you combine Cycle **224** with one of the machining cycles, the **Safety clearance**, coordinate surface and 2nd set-up clearance that you defined in Cycle **224** will be effective for the selected machining cycle. There is a danger of collision!

- Check the machining sequence using a graphic simulation
- Carefully test the NC program or program section in **SINGLE BLOCK** mode of **Program run** operating mode.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **224** is DEF-active. In addition, Cycle **224** automatically calls the last defined machining cycle.
- The control uses the special character **%** for special functions. If you want to use this character in a DataMatrix code, enter it twice in the text (e.g., **%%**).

Cycle parameters

Help graphic	Parameter
	**Q225 Starting point in 1st axis?** Coordinate in the lower left corner of the code in the main axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Coordinate in the bottom left corner of the data matrix code in the secondary axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	QS501 Text input? Enter the text to be converted within quotation marks. Variables can be assigned. **Further information:** "Outputting variable texts in DataMatrix codes", Page 442 Input: Max. **255** characters
	Q458 Cell size/Pattern size(1/2)? Specify how the DataMatrix code is described in **Q459**: **1**: Distance between cells **2**: Pattern size Input: **1**, **2**
	**Q459 Size for pattern?** Definition of the distance between cells or the pattern size: If **Q458=1**: Distance between the first and second cell (between cell centers) If **Q458=2**: Distance between the first and last cell (between cell centers) This value has an incremental effect. Input: **0...99999.9999**
	**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** or **PREDEF**
	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**

Help graphic	Parameter
	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** or **PREDEF**

Example

```
11 CYCL DEF 224 DATAMATRIX CODE PATTERN ~
    Q225=+0                ;STARTNG PNT 1ST AXIS ~
    Q226=+0                ;STARTNG PNT 2ND AXIS ~
    QS501=""               ;TEXT ~
    Q458=+1                ;SIZE SELECTION ~
    Q459=+1                ;SIZE ~
    Q224=+0                ;ANGLE OF ROTATION ~
    Q200=+2                ;SET-UP CLEARANCE ~
    Q203=+0                ;SURFACE COORDINATE ~
    Q204=+50               ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

Outputting variable texts in DataMatrix codes

In addition to specified characters you can also output certain variables in DataMatrix codes. Precede the variable with **%**.

You can use the following variable texts in Cycle **224 DATAMATRIX CODE PATTERN**:

- Date and time
- Names and paths of NC programs
- Count values

Date and time

You can convert the current date, the current time, or the current calendar week into a DataMatrix code. Enter the value **%time<x>** in cycle parameter **QS501**. **<x>** defines the format (e.g., 08 for DD.MM.YYYY.)

> Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9 (e.g., **%time08**).

The following formats are available:

Input	Format
%time00	DD.MM.YYYY hh:mm:ss
%time01	D.MM.YYYY h:mm:ss
%time02	D.MM.YYYY h:mm
%time03	D.MM.YY h:mm
%time04	YYYY-MM-DD hh:mm:ss
%time05	YYYY-MM-DD hh:mm
%time06	YYYY-MM-DD h:mm
%time07	YY-MM-DD h:mm
%time08	DD.MM.YYYY
%time09	D.MM.YYYY
%time10	D.MM.YY
%time11	YYYY-MM-DD
%time12	YY-MM-DD
%time13	hh:mm:ss
%time14	h:mm:ss
%time15	h:mm
%time99	Calendar week

Names and paths of NC programs

You can convert the name or path of the active or called NC program into a DataMatrix code. Enter the value **%main<x>** or **%prog<x>** in cycle parameter **QS501**.

The following formats are available:

Input	Meaning	Example
%main0	Full path of the active NC program	**TNC:\MILL.h**
%main1	Directory path of the active NC program	**TNC:**
%main2	Name of the active NC program	**MILL**
%main3	File type of the active NC program	**.H**
%prog0	Full path of the called NC program	**TNC:\HOUSE.h**
%prog1	Directory path of the called NC program	**TNC:**
%prog2	Name of the called NC program	**HOUSE**
%prog3	File type of the called NC program	**.H**

Count values

You can convert the current count value into a DataMatrix code. The control displays the current count value in **Program Run** on the**PGM** tab of the **Status** workspace.

Enter the value **%count<x>** in cycle parameter **QS501**.

The number after **%count** indicates how many digits the DataMatrix code contains. The maximum is nine digits.

Example:

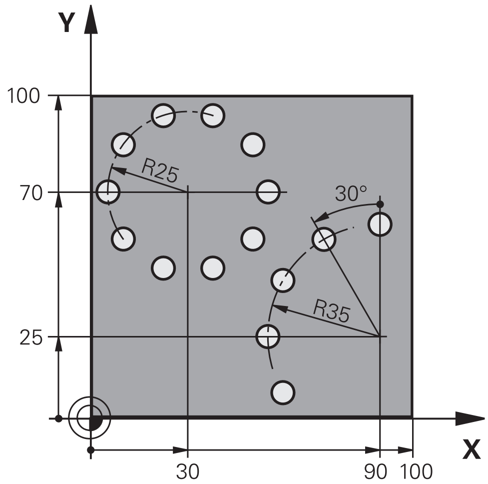
- Programming: **%count9**
- Current count value: 3
- Result: 000000003

Operating information

- In the Simulation, the control only simulates the count value you define directly in the NC program. The count value from the **Status** workspace in the **Program Run** operating mode is ignored.

14.6.5 Programming Examples

Example: Polar hole patterns



0 BEGIN PGM 200 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 TOOL CALL 200 Z S3500	; Tool call
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-15 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+4 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0.25 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
6 CYCL DEF 220 POLAR PATTERN ~	
Q216=+30 ;CENTER IN 1ST AXIS ~	
Q217=+70 ;CENTER IN 2ND AXIS ~	
Q244=+50 ;PITCH CIRCLE DIAMETR ~	
Q245=+0 ;STARTING ANGLE ~	
Q246=+360 ;STOPPING ANGLE ~	
Q247=+0 ;STEPPING ANGLE ~	
Q241=+10 ;NR OF REPETITIONS ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+100 ;2ND SET-UP CLEARANCE ~	
Q301=+1 ;MOVE TO CLEARANCE ~	
Q365=+0 ;TYPE OF TRAVERSE	

7 CYCL DEF 220 POLAR PATTERN ~	
Q216=+90 ;CENTER IN 1ST AXIS ~	
Q217=+25 ;CENTER IN 2ND AXIS ~	
Q244=+70 ;PITCH CIRCLE DIAMETR ~	
Q245=+90 ;STARTING ANGLE ~	
Q246=+360 ;STOPPING ANGLE ~	
Q247=+30 ;STEPPING ANGLE ~	
Q241=+5 ;NR OF REPETITIONS ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+100 ;2ND SET-UP CLEARANCE ~	
Q301=+1 ;MOVE TO CLEARANCE ~	
Q365=+0 ;TYPE OF TRAVERSE	
8 L Z+100 R0 FMAX	; Retract the tool
9 M30	; End of program
10 END PGM 200 MM	

14.7 OCM cycles for pattern definition

14.7.1 Overview

OCM figures

Cycle	Call	Further information
1271 OCM RECTANGLE (option 167) ■ Definition of a rectangle ■ Input of the side lengths ■ Definition of the corners	**DEF-**active	Page 449
1272 OCM CIRCLE (option 167) ■ Definition of a circle ■ Input of the circle diameter	**DEF-**active	Page 452
1273 OCM SLOT / RIDGE (option 167) ■ Definition of a slot or ridge ■ Input of the width and the length	**DEF-**active	Page 454
1278 OCM POLYGON (option 167) ■ Definition of a polygon ■ Input of the reference circle ■ Definition of the corners	**DEF-**active	Page 458
1281 OCM RECTANGLE BOUNDARY (option 167) ■ Definition of a bounding rectangle	**DEF-**active	Page 461
1282 OCM CIRCLE BOUNDARY (option 167) ■ Definition of a bounding circle	**DEF-**active	Page 463

14.7.2 Fundamentals

The control provides cycles for frequently used figures. You can program these figures as pockets, islands, or boundaries.

These figure cycles offer the following benefits:

- You can conveniently program the figures and machining data without the need to program an individual path contour.
- Frequently needed figures can be reused.
- If you want to program an island or an open pocket, the control provides you with more cycles for defining the figure boundary.
- The Boundary figure type enables you to face-mill your figure.

With a figure, you can redefine the OCM contour data and cancel the definition of a previously defined Cycle **271 OCM CONTOUR DATA** or of a figure boundary.

For defining figures, the control provides the following cycles:

- **1271 OCM RECTANGLE**, see Page 449
- **1272 OCM CIRCLE**, see Page 452
- **1273 OCM SLOT / RIDGE**, see Page 454
- **1278 OCM POLYGON**, see Page 458

For defining figure boundaries, the control provides the following cycles:

- **1281 OCM RECTANGLE BOUNDARY**, see Page 461
- **1282 OCM CIRCLE BOUNDARY**, see Page 463

Tolerances

The control allows you to store tolerances in the following cycles and cycle parameters:

Cycle number	Parameter
1271 OCM RECTANGLE	**Q218 FIRST SIDE LENGTH**, **Q219 2ND SIDE LENGTH**
1272 OCM CIRCLE	**Q223 CIRCLE DIAMETER**
1273 OCM SLOT / RIDGE	**Q219 SLOT WIDTH**, **Q218 SLOT LENGTH**
1278 OCM POLYGON	**Q571 REF-CIRCLE DIAMETER**

You can define the following tolerances:

Tolerances	Example	Manufacturing dimension
Dimensions	10+0.01-0.015	9.9975
DIN EN ISO 286-2	10H7	10.0075
DIN ISO 2768-1	10m	10.0000

> Pay attention to capitalization when entering tolerances.

Proceed as follows:

- Start the cycle definition
- Define the cycle parameters
- Select **TEXT** in the action bar
- Enter a nominal dimension including tolerance

> If you program an incorrect tolerance, the control interrupts machining with an error message.

14.7.3 Cycle 1271 OCM RECTANGLE (option 167)

ISO programming
G1271

Application

Use the figure cycle **1271 OCM RECTANGLE** to program a rectangle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program tolerances for the lengths.

If you work with Cycle **1271**, program the following:

- Cycle **1271 OCM RECTANGLE**
  - If you program **Q650**=1 (figure type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

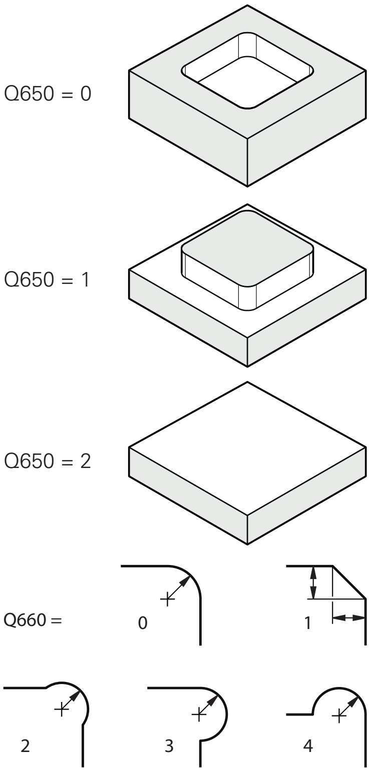
Notes

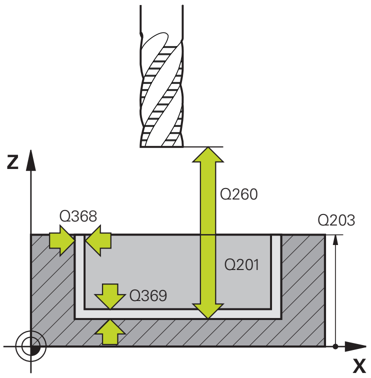
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1271** are valid for the OCM machining cycles **272** to **274** and **277**.

Notes on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you want to machine a figure at several positions after initial roughing, then program the number or name of the rough-out tool in the OCM machining cycle. If there was no initial roughing, you need to define **Q438 = 0** in the cycle for the first roughing operation.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q218 First side length? Length of the first side of the figure, parallel to the main axis. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 448 Input: **0...99999.9999**
	Q219 Second side length? Length of the 2nd side of the figure, parallel to the secondary axis. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 448 Input: **0...99999.9999**
	Q660 Type of corners? Geometry of the corners: **0**: Radius **1**: Chamfer **2**: Milling corners in the main and secondary axis directions **3**: Milling corners in the main axis direction **4**: Milling corners in the secondary axis direction Input: **0**, **1**, **2**, **3**, **4**
	Q220 Corner radius? Radius or chamfer of the corner of the figure Input: **0...99999.9999**
	Q367 Position of pocket (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 = 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**
	Q224 Angle of rotation? Angle by which the figure is rotated. The center of rotation is at the center of the figure. This value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**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**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners? The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1271 OCM RECTANGLE ~
    Q650=+1          ;FIGURE TYPE ~
    Q218=+60         ;FIRST SIDE LENGTH ~
    Q219=+40         ;2ND SIDE LENGTH ~
    Q660=+0          ;CORNER TYPE ~
    Q220=+0          ;CORNER RADIUS ~
    Q367=+0          ;POCKET POSITION ~
    Q224=+0          ;ANGLE OF ROTATION ~
    Q203=+0          ;SURFACE COORDINATE ~
    Q201=-10         ;DEPTH ~
    Q368=+0          ;ALLOWANCE FOR SIDE ~
    Q369=+0          ;ALLOWANCE FOR FLOOR ~
    Q260=+50         ;CLEARANCE HEIGHT ~
    Q578=+0.2        ;INSIDE CORNER FACTOR
```

14.7.4 Cycle 1272 OCM CIRCLE (option 167)

ISO programming
G1272

Application

Use figure cycle **1272 OCM CIRCLE** to program a circle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the diameter.

If you work with Cycle **1272**, program the following:

- Cycle **1272 OCM CIRCLE**
  - If you program **Q650**=1 (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

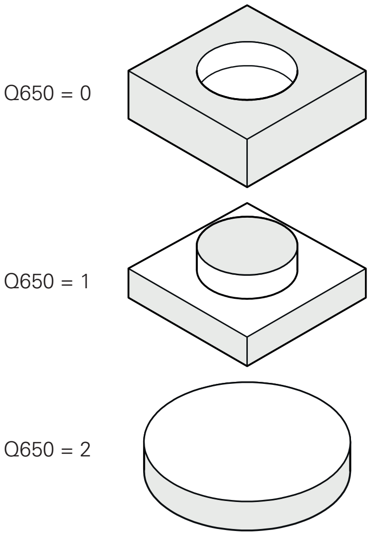
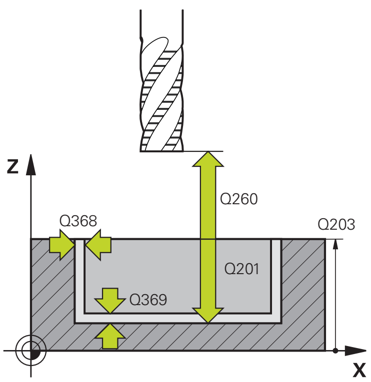
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1272** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1272** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you want to machine a figure at several positions after initial roughing, then program the number or name of the rough-out tool in the OCM machining cycle. If there was no initial roughing, you need to define **Q438 = 0** in the cycle for the first roughing operation.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q223 Circle diameter? Diameter of the finished circle. You can program a tolerance if needed. **Further information:** "Tolerances", Page 448 Input: **0...99999.9999**
	Q367 Position of pocket (0/1/2/3/4)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool pos. = Center of figure **1**: Tool pos. = Quadrant transition at 90° **2**: Tool pos. = Quadrant transition at 0° **3**: Tool pos. = Quadrant transition at 270° **4**: Tool pos. = Quadrant transition at 180° Input: **0**, **1**, **2**, **3**, **4**
	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**
	**Q201 Depth?** Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q578 Radius factor on inside corners? The minimum radius of a circular pocket results from the tool radius plus the product of tool radius and **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1272 OCM CIRCLE ~
    Q650=+0             ;FIGURE TYPE ~
    Q223=+50            ;CIRCLE DIAMETER ~
    Q367=+0             ;POCKET POSITION ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q201=-20            ;DEPTH ~
    Q368=+0             ;ALLOWANCE FOR SIDE ~
    Q369=+0             ;ALLOWANCE FOR FLOOR ~
    Q260=+100           ;CLEARANCE HEIGHT ~
    Q578=+0.2           ;INSIDE CORNER FACTOR
```

14.7.5 Cycle 1273 OCM SLOT / RIDGE (option 167)

ISO programming
G1273

Application

Use figure cycle **1273 OCM SLOT / RIDGE** to program a slot or a ridge. This figure cycle also allows you to program a boundary for face milling. In addition, you can program a tolerance for the width and the length.

If you work with Cycle **1273**, program the following:

- Cycle **1273 OCM SLOT / RIDGE**
  - If you program **Q650**=1 (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

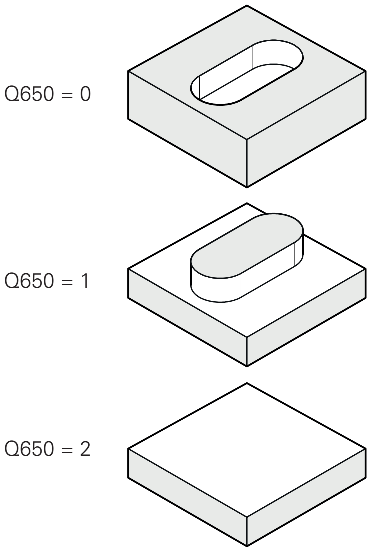
Notes

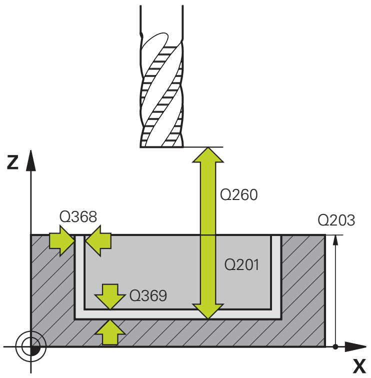
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1273** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1273** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you want to machine a figure at several positions after initial roughing, then program the number or name of the rough-out tool in the OCM machining cycle. If there was no initial roughing, you need to define **Q438 = 0** in the cycle for the first roughing operation.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q219 Width of slot? Width of the slot or ridge, parallel to the secondary axis of the working plane. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 448 Input: **0...99999.9999**
	Q218 Length of slot? Length of the slot or ridge, parallel to the main axis of the working plane. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 448 Input: **0...99999.9999**
	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**
	Q224 Angle of rotation? Angle by which the figure is rotated. The center of rotation is at the center of the figure. This value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**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**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners? The minimum radius (slot width) of a slot results from the tool radius plus the product of tool radius and **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1273 OCM SLOT / RIDGE ~
    Q650=+0              ;FIGURE TYPE ~
    Q219=+10             ;SLOT WIDTH ~
    Q218=+60             ;SLOT LENGTH ~
    Q367=+0              ;SLOT POSITION ~
    Q224=+0              ;ANGLE OF ROTATION ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q201=-20             ;DEPTH ~
    Q368=+0              ;ALLOWANCE FOR SIDE ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q578=+0.2            ;INSIDE CORNER FACTOR
```

14.7.6 Cycle 1278 OCM POLYGON (option 167)

ISO programming

G1278

Application

Use figure cycle **1278 OCM POLYGON** to program a polygon. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the reference diameter.

If you work with Cycle **1278**, program the following:

- Cycle **1278 OCM POLYGON**
  - If you program **Q650**=1 (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

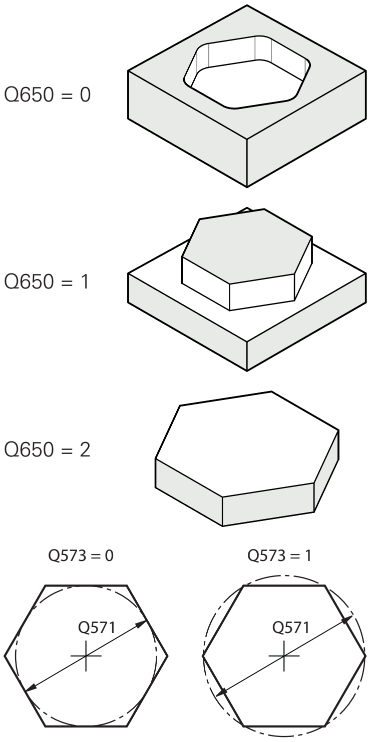
Notes

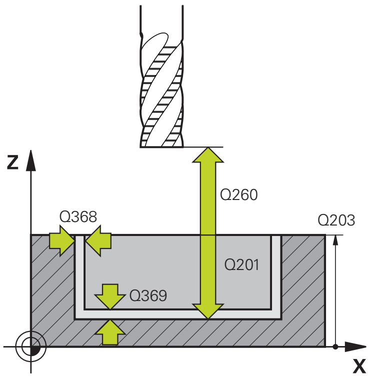
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1278** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1278** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you want to machine a figure at several positions after initial roughing, then program the number or name of the rough-out tool in the OCM machining cycle. If there was no initial roughing, you need to define **Q438 = 0** in the cycle for the first roughing operation.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q573 Inscr.circle/circumcircle (0/1)? Define whether the dimension **Q571** is referenced to the inscribed circle or the circumcircle: **0**: Dimension is referenced to the inscribed circle **1**: Dimension is referenced to the circumcircle Input: **0**, **1**
	Q571 Reference circle diameter? Enter the diameter of the reference circle. Specify in parameter **Q573** whether the diameter entered here is referenced to the inscribed circle or the circumcircle. You can program a tolerance if needed. **Further information:** "Tolerances", Page 448 Input: **0...99999.9999**
	Q572 Number of corners? Enter the number of corners of the polygon. The control will always distribute the corners evenly on the polygon. Input: **3...30**
	Q660 Type of corners? Geometry of the corners: **0**: Radius **1**: Chamfer Input: **0**, **1**
	Q220 Corner radius? Radius or chamfer of the corner of the figure Input: **0...99999.9999**
	Q224 Angle of rotation? Angle by which the figure is rotated. The center of rotation is at the center of the figure. This value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**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**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners? The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1278 OCM POLYGON ~
    Q650=+0          ;FIGURE TYPE ~
    Q573=+0          ;REFERENCE CIRCLE ~
    Q571=+50         ;REF-CIRCLE DIAMETER ~
    Q572=+6          ;NUMBER OF CORNERS ~
    Q660=+0          ;CORNER TYPE ~
    Q220=+0          ;CORNER RADIUS ~
    Q224=+0          ;ANGLE OF ROTATION ~
    Q203=+0          ;SURFACE COORDINATE ~
    Q201=-10         ;DEPTH ~
    Q368=+0          ;ALLOWANCE FOR SIDE ~
    Q369=+0          ;ALLOWANCE FOR FLOOR ~
    Q260=+50         ;CLEARANCE HEIGHT ~
    Q578=+0.2        ;INSIDE CORNER FACTOR
```

14.7.7 Cycle 1281 OCM RECTANGLE BOUNDARY (option 167)

ISO programming
G1281

Application

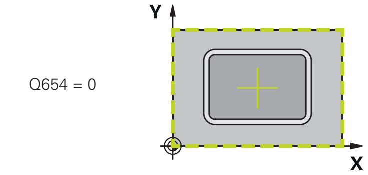
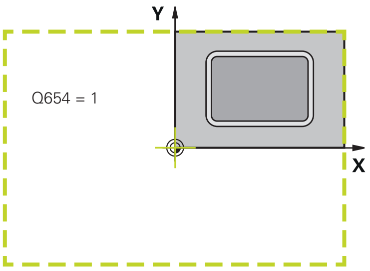
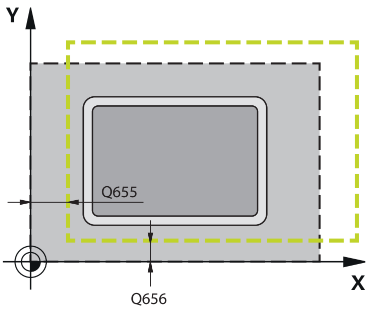
Use Cycle **1281 OCM RECTANGLE BOUNDARY** to program a rectangular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard figure.

The cycle becomes effective when you program cycle parameter **Q650 FIGURE TYPE** = 0 (pocket) or = 1 (island) within an OCM standard figure cycle.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1281** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1281** are valid for Cycles **1271** to **1273** and **1278**.

Cycle parameters

Help graphic	Parameter
	**Q651 Length of major axis?** Length of the first side of the boundary, parallel to the main axis Input: **0.001...9999.999**
	Q652 Length of minor axis? Length of the second side of the boundary, parallel to the secondary axis Input: **0.001...9999.999**
	**Q654 Position reference for figure?** Specify the position reference for the center: **0**: The center of the boundary is referenced to the center of the contour **1**: The center of the boundary is referenced to the datum Input: **0**, **1**
	**Q655 Shift in major axis?** Shift of the rectangle boundary along the main axis Input: **-999.999...+999.999**
	Q656 Shift in minor axis? Shift of the rectangle boundary along the secondary axis Input: **-999.999...+999.999**

Example

```
11 CYCL DEF 1281 OCM RECTANGLE BOUNDARY ~
    Q651=+50                ;LENGTH 1 ~
    Q652=+50                ;LENGTH 2 ~
    Q654=+0                 ;POSITION REFERENCE ~
    Q655=+0                 ;SHIFT 1 ~
    Q656=+0                 ;SHIFT 2
```

14.7.8 Cycle 1282 OCM CIRCLE BOUNDARY (option 167)

ISO programming

G1282

Application

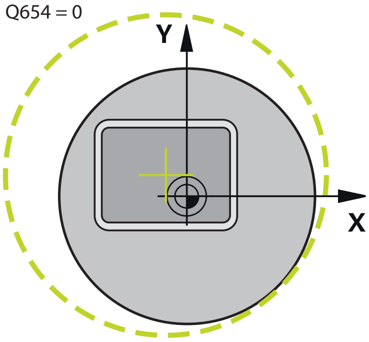
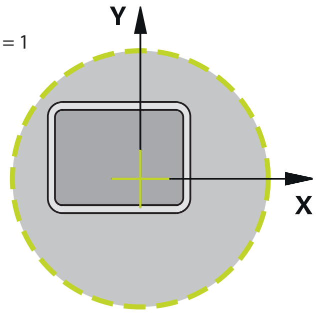
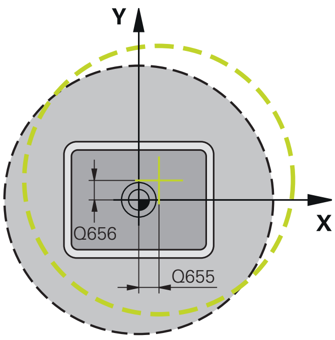
Cycle **1282 OCM CIRCLE BOUNDARY** allows you to program a circular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard figure.

The cycle becomes effective when you program cycle parameter **Q650 FIGURE TYPE** = **0** (pocket) or = **1** (island) in an OCM standard shape cycle.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1282** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1282** are valid for Cycles **1271** to **1273** and **1278**.

Cycle parameters

Help graphic	Parameter
Q654 = 0  	**Q653 Diameter?** Diameter of the circular bounding frame Input: **0.001...9999.999**
	Q654 Position reference for figure? Specify the position reference for the center: **0**: The center of the boundary is referenced to the center of the contour **1**: The center of the boundary is referenced to the datum Input: **0**, **1**
Q654 = 1  	**Q655 Shift in major axis?** Shift of the rectangle boundary along the main axis Input: **-999.999...+999.999**
	Q656 Shift in minor axis? Shift of the rectangle boundary along the secondary axis Input: **-999.999...+999.999**
 	

Example

11 CYCL DEF 1282 OCM CIRCLE BOUNDARY ~	
Q653=+50	**;DIAMETER ~**
Q654=+0	**;POSITION REFERENCE ~**
Q655=+0	**;SHIFT 1 ~**
Q656=+0	**;SHIFT 2**

14.8 Recessing and undercutting

14.8.1 Recessing and undercutting

Some cycles machine contours that you have written in a subprogram. Further special contour elements are available to you for writing turning contours. In this way you can program recessing and undercutting as complete contour elements with a single NC block.

> Recessing and undercutting are always referenced to a previously defined linear contour element.
>
> You can only use the recess and undercut elements GRV and UDC in contour subprograms that have been called by a turning cycle.

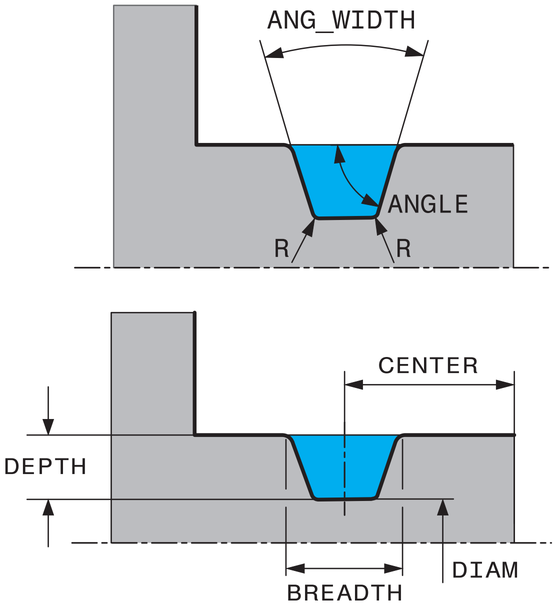
Various input options are available to you for defining undercuts and recesses. Some of these inputs have to be made (mandatory input); others can be skipped (optional input). The mandatory inputs are symbolized as such in the help graphics. In some elements, you can select between two different definitions. The control provides relevant selection possibilities via an action bar.

The control provides various possibilities for programming recesses and undercuts in the **Recess / Undercut** folder of the **Insert NC function** window.

Programming recessing

Recessing is the machining of recesses into round parts, usually for accommodation of locking rings and seals, or as lubricating grooves. You can program recessing around the circumference or on the face end of the turned part. You have two separate contour elements for this purpose:

- **GRV RADIAL**: Recess in circumference of component
- **GRV AXIAL**: Recess on face end of component



Input parameters in recessing GRV

Parameter	Meaning	Input
CENTER	Center of recess	Required
R	Corner radius of both inside corners	Optional
DEPTH / **DIAM**	Depth of recess (pay attention to algebraic sign!) /diameter of recess base	Required
BREADTH	Recess width	Required
ANGLE / **ANG_WIDTH**	Flank angle / opening angle between both flanks	Optional
RND / **CHF**	Rounding / chamfer on contour corner near to starting point	Optional
FAR_RND / **FAR_CHF**	Rounding / chamfer on contour corner away from starting point	Optional

> The algebraic sign for the recess depth specifies the machining position (inside/outside machining) of the recess.
> Algebraic signs of recess depth for outside machining:
> - If the contour element is in the negative direction of the Z coordinate, use a negative sign
> - If the contour element is in the positive direction of the Z coordinate, use a positive sign
>
> Algebraic signs of recess depth for inside machining:
> - If the contour element is in the negative direction of the Z coordinate, use a positive sign
> - If the contour element is in the positive direction of the Z coordinate, use a negative sign

Example: Radial recess with depth=5, width=10, pos.= Z-15

```
11 L  X+40  Z+0
12 L  Z-30
13 GRV RADIAL CENTER-15 DEPTH-5 BREADTH10 CHF1 FAR_CHF1
14 L  X+60
```

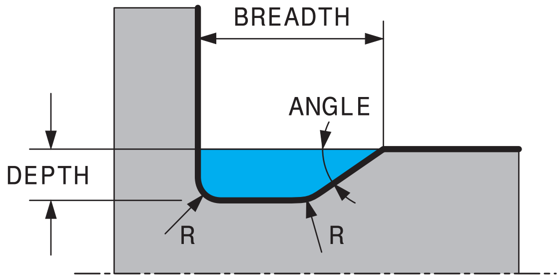
Programming undercutting

Undercutting is usually required for the flush connection of components. In addition, undercutting can help reduce the notch effect at corners. Threads and fits are often machined with an undercut. You have various contour elements for defining the different undercuts:

- **UDC TYPE_E**: Undercut for cylindrical surfaces to be further processed as per DIN 509.
- **UDC TYPE_F**: Undercut for plane surface and cylindrical surface to be further processed as per DIN 509
- **UDC TYPE_H**: Undercut for more rounded transition as per DIN 509
- **UDC TYPE_K**: Undercut in plane surface and cylindrical surface
- **UDC TYPE_U**: Undercut in cylindrical surface
- **UDC THREAD**: Thread undercut as per DIN 76

> The control always interprets undercuts as form elements in the longitudinal direction. No undercuts are possible in the plane direction.

Undercut DIN 509 UDC TYPE _E



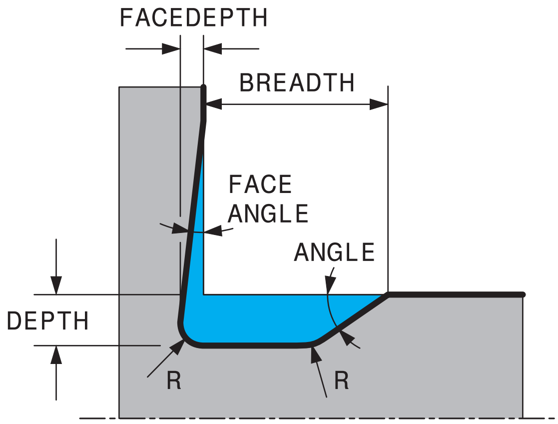
Input parameters in undercut DIN 509 UDC TYPE_E

Parameter	Meaning	Input
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Undercut with depth = 2, width = 15

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_E R1 DEPTH2 BREADTH15
14 L  X+60
```

Undercut DIN 509 UDC TYPE_F



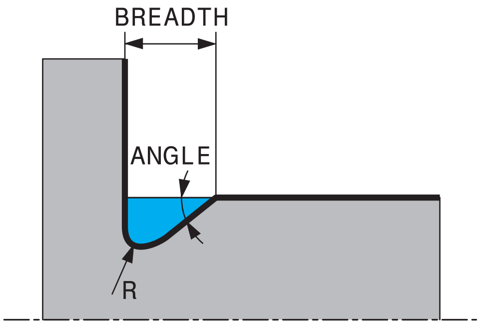
Input parameters in undercut DIN 509 UDC TYPE_F

Parameter	Meaning	Input
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional
FACEDEPTH	Depth of face	Optional
FACEANGLE	Contour angle of face	Optional

Example: Undercut form F with depth = 2, Width = 15, Depth of face = 1

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_F R1 DEPTH2 BREADTH15 FACEDEPTH1
14 L  X+60
```

Undercut DIN 509 UDC TYPE_H



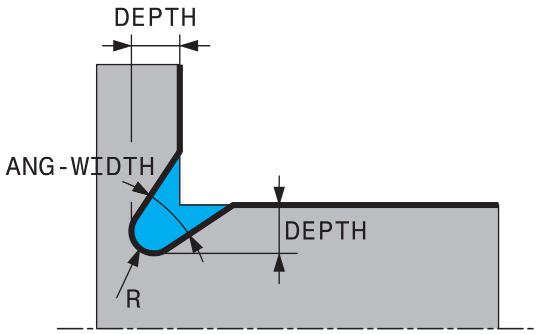
Input parameters in undercut DIN 509 UDC TYPE_H

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
BREADTH	Width of undercut	Required
ANGLE	Undercut angle	Required

Example: Undercut form H with depth = 2, width = 15, angle = 10°

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_H R1 BREADTH10 ANGLE10
14 L  X+60
```

Undercut UDC TYPE_K



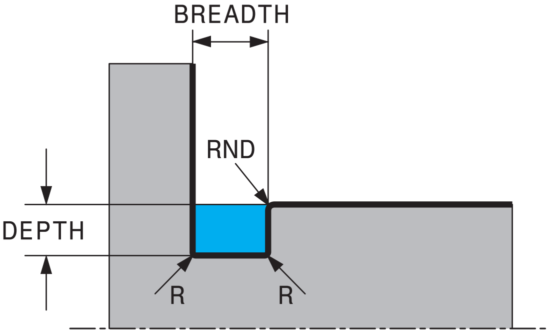
Input parameters in undercut UDC TYPE_K

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
DEPTH	Undercut depth (parallel to axis)	Required
ROT	Angle relative to longitudinal axis (default: 45°)	Optional
ANG_WIDTH	Angle of undercut opening	Required

Example: Undercut form K with depth = 2, width = 15, opening angle = 30°

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_K R1 DEPTH3 ANG_WIDTH30
14 L  X+60
```

Undercut UDC TYPE_U



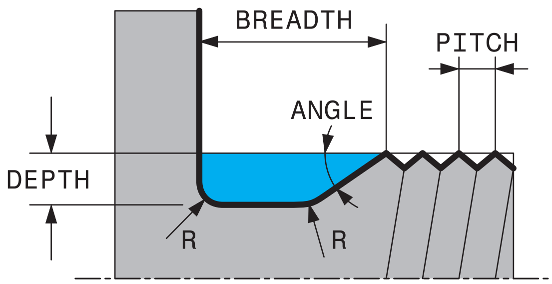
Input parameters in undercut UDC TYPE_U

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
DEPTH	Undercut depth	Required
BREADTH	Width of undercut	Required
RND / **CHF**	Rounding / chamfer on outside corner	Required

Example: Undercut form U with depth = 3, width = 8

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_U R1 DEPTH3 BREADTH8 RND1
14 L  X+60
```

Undercut UDC THREAD



Input parameters in undercut DIN 76 UDC THREAD

Parameter	Meaning	Input
PITCH	Thread pitch	Optional
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Thread undercut according to DIN 76 with thread pitch = 2

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC THREAD PITCH2
14 L  X+60
```

15

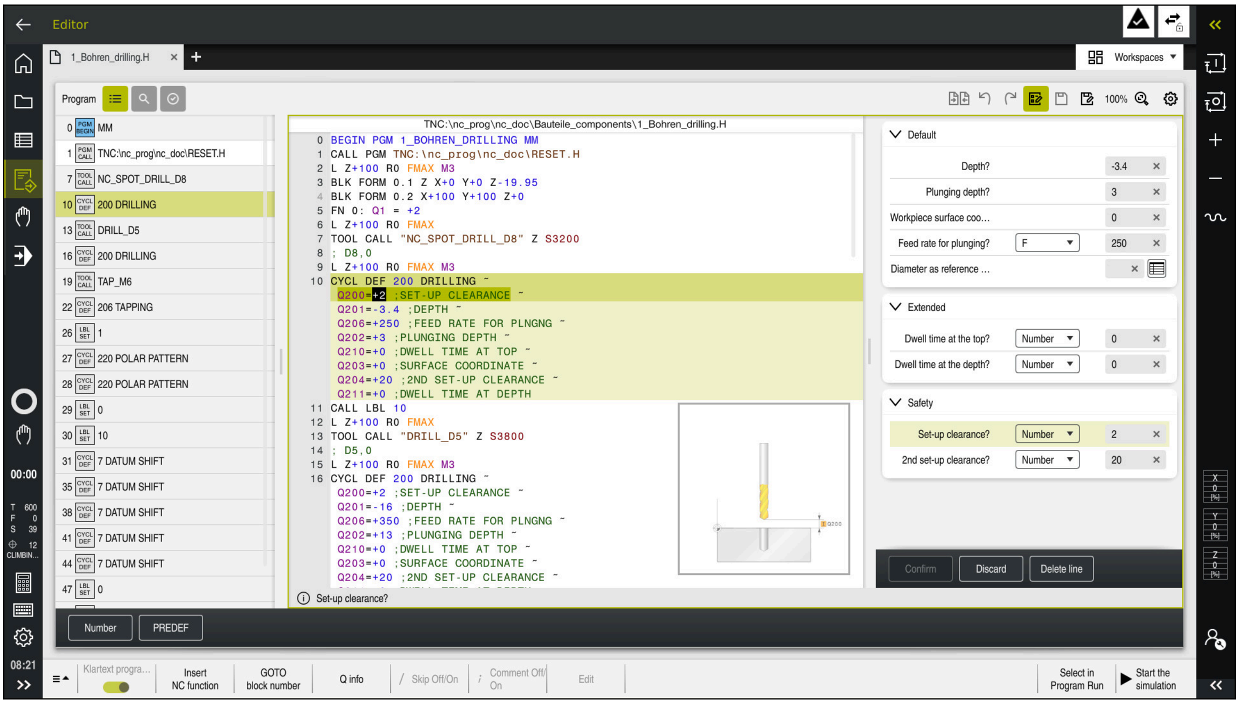
Machining Cycles

15.1 Working with machining cycles

15.1.1 Machining cycles

> 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.

General information



Cycles are stored on the control as subprograms. The cycles can be used to execute different machining operations. This greatly simplifies the task of creating programs. The cycles are also useful for frequently recurring machining operations that comprise several working steps. Most cycles use Q parameters as transfer parameters. The control provides cycles for the following technologies:

- Drilling processes
- Thread machining
- Milling operations such as pockets, studs or even contours
- Cycles for coordinate transformation
- Special cycles
- Turning operations
- Grinding operations

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

- Simulate your program before executing it

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

Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, which were not all available in some older software versions. Within a cycle, these parameters are always provided at the end of the cycle definition. The section "New functions 81762x-17" gives you an overview of the optional Q parameters that have been added in this software version. You can decide for yourself whether you would like to define optional Q parameters or delete them with the **NO ENT** key. You can also adopt the default value. If you have accidentally deleted an optional Q parameter or if you would like to extend cycles in your existing NC programs, you can add optional Q parameters in cycles where needed. The following steps describe how this is done.

Proceed as follows:

- Call the cycle definition
- Press the right arrow key until the new Q parameters are displayed
- Confirm the displayed default value

  or
- Enter a value
- To load the new Q parameter, exit the menu by selecting the right arrow key once again or by selecting the **END** button
- If you do not wish to load the new Q parameter, press the **NO ENT** key

Compatibility

Most NC programs created with older HEIDENHAIN controls (as of TNC 150 B) can be run with the new software version of the TNC7. Even if new optional parameters have been added to existing cycles, you will generally be able to run your NC programs as usual. This is achieved because the stored default value will be used. The other way around, if you want to run an NC program created with a new software version on an older control, you can delete the respective optional Q parameters from the cycle definition with the **NO ENT** key. In this way you can ensure that the NC program is downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

15.1.2 Defining cycles

Cycles can be defined in several ways.

Inserting via NC function:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Inserting via the CYCL DEF key:

CYCL DEF

- ▶ Press the **CYCL DEF** key
- > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Navigation in the cycle

Key	Function
▶	Navigation within the cycle: Jump to next parameter
◀	Navigation within the cycle: Jump to previous parameter
▼	Jump to the same parameter in the next cycle
▲	Jump to the same parameter in the previous cycle

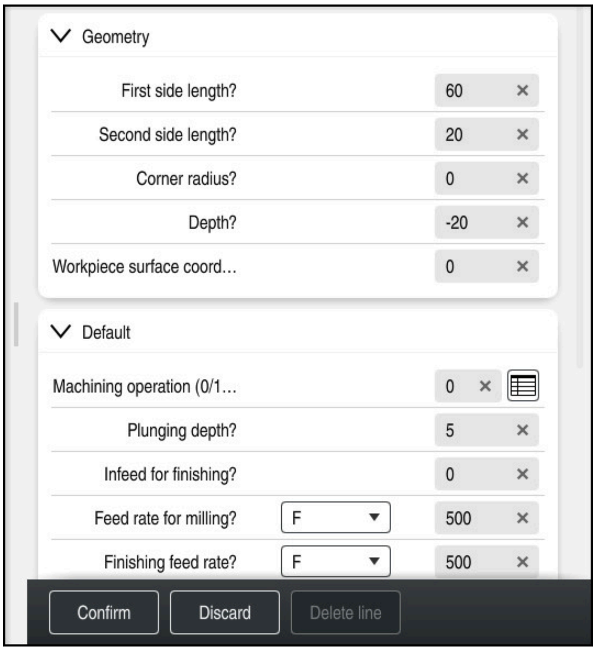
The control provides selection possibilities for the different cycle parameters via the action bar or the form.

If an input option specifying a defined behavior is stored in particular cycle parameters, you can open a selection list with the **GOTO** key or in the form view. For example in cycle **200 DRILLING**, the **Q395 DEPTH REFERENCE** parameter provides the selection possibility:

- 0 | Tool tip
- 1 | Cutting edge corner

Cycle input form

The control provides a **FORM** for various functions and cycles. This **FORM** allows you to enter various syntax elements or cycle parameters.



The control allocates the cycle parameters in the **FORM** to groups based on their functions, e.g. geometry, standard, advanced, safety. The control provides selection possibilities for different cycle parameters via switches, for example. The control displays the currently edited cycle parameter in color.

After you have defined all required cycle parameters, you can confirm your input and conclude the cycle.

Opening the form:

- Open the **Editor** operating mode
- Open the **Program** workspace

- Select **FORM** via the title bar

If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

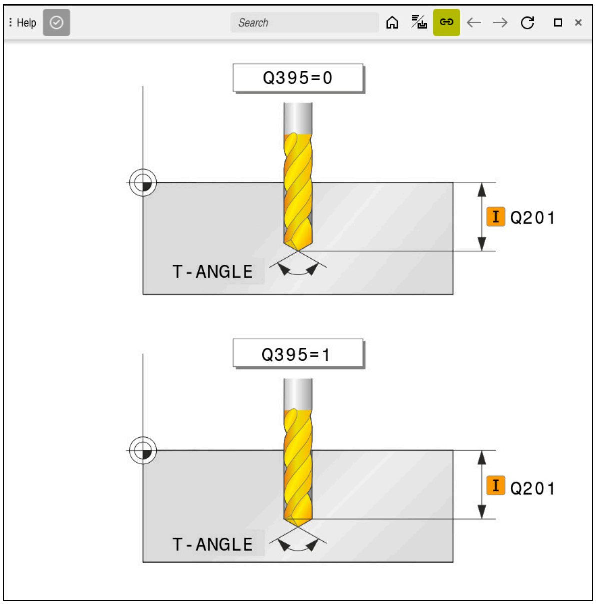
Help graphics

When you are editing a cycle, the control shows a help graphic for the current Q parameters. The size of the help graphic depends on the size of the **Program** workspace area.

The control shows the help graphic at the right edge of the workspace, or at the top or bottom edge. The help graphic is positioned in the half that does not contain the cursor.

When you tap or click on the help graphic, the control maximizes the help graphic.

If the **Help** workspace is active, then the control displays the help graphic in it rather than in the **Program** workspace.



Help workspace with a help graphic for a cycle parameter

15.1.3 Calling cycles

For cycles that remove material, you have to enter not only the cycle definition, but also the cycle call in the NC program. The call always refers to the fixed cycle that was last defined in the NC program.

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (only required for simulation)
- Tool call
- Spindle direction of rotation (miscellaneous function **M3**/**M4**)
- Cycle definition (**CYCL DEF**)

> - For some cycles, additional requirements must be observed. They are detailed in the descriptions and overview tables for each cycle.

You can program the cycle call in the following ways.

Option	Further information
CYCL CALL	Page 481
CYCL CALL PAT	Page 481
CYCL CALL POS	Page 482
M89/M99	Page 482

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.

Insert NC function

- Select **Insert NC function**

or

CYCL CALL

- Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.
- Select **CYCL CALL M**
- Define **CYCL CALL M** and add an M function , if necessary

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 419

Further information: "Point tables", Page 400

Insert NC function

- Select **Insert NC function**

or

CYCL CALL

- Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.
- Select **CYCL CALL PAT**
- Define **CYCL CALL PAT** and add an M function , if necessary

Calling a cycle with CYCL CALL POS

The **CYCL CALL POS** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that you defined in the **CYCL CALL POS** block.

Insert NC function

- ▶ Select **Insert NC function**

  or

CYCL CALL

- ▶ Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.
- ▶ Select **CYCL CALL POS**
- ▶ Define **CYCL CALL POS** and add an M function , if necessary

Using positioning logic, the control moves to the position defined in the **CYCL CALL POS** block:

- If the tool's current position in the tool axis is above the upper edge of the workpiece (**Q203**), the control first moves the tool to the programmed position in the working plane and then to the programmed position in the tool axis
- If the tool's current position in the tool axis is below the upper edge of the workpiece (**Q203**), the control first moves the tool to the clearance height in the tool axis and then to the programmed position in the working plane

> Programming and operating notes
>
> - Three coordinate axes must always be programmed in the **CYCL CALL POS** block. Using the coordinate in the tool axis, you can easily change the starting position. It serves as an additional datum shift.
> - The feed rate most recently defined in the **CYCL CALL POS** block is only used to traverse to the start position programmed in this block.
> - As a rule, the control moves without radius compensation (R0) to the position defined in the **CYCL CALL POS** block.
> - If you use **CYCL CALL POS** to call a cycle in which a start position is defined (e.g., Cycle **212**), then the position defined in the cycle serves as an additional shift of the position defined in the **CYCL CALL POS** block. You should therefore always define the start position in the cycle as 0.

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**, proceed as follows:

- ▶ Program **M99** in the positioning block
- > The control moves to the last starting point.

  or

- ▶ Define a new machining cycle with **CYCL DEF**

Defining and calling an NC program as cycle

With **SEL CYCLE**, you can define any NC program as a machining cycle.

Defining an NC program as a cycle:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **SEL CYCLE**
- Select file name, string parameter or file

Calling an NC program as a cycle:

CYCL CALL

- Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.

  or
- Program **M99**

- 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.
- Please note that **CYCL CALL PAT** and **CYCL CALL POS** use positioning logic before executing the cycle. With respect to the positioning logic, **SEL 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 at the starting point of the pattern
  - 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.

15.1.4 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:

Cycle-number range	Description
300 to **399**	Machine-specific cycles that are to be selected through the **CYCL DEF** key
500 to **599**	Machine-specific touch probe cycles that are to be selected through the **TOUCH PROBE** 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 cycles", Page 481
Further information: "Variables: Q, QL, QR and QS parameters", Page 1362

15.1.5 Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread	
■ Drilling, reaming	Page 488
■ Boring	Page 507
■ Counterboring, centering	
■ Tapping or thread milling	
Pockets/studs/slots	
■ Pocket milling	Page 507
■ Stud milling	
■ Slot milling	
■ Face milling	
Coordinate transformations	
■ Mirroring	Page 1034
■ Rotating	
■ Magnifying / Reducing	
SL cycles	
■ SL (Subcontour List) cycles for the machining of contours that possibly consist of several subcontours	Page 507
■ Cylinder surface machining	Page 1266
■ OCM (Optimized Contour Milling) cycles for combining subcontours to form complex contours	Page 446
Point patterns	
■ Bolt hole circle	Page 431
■ Linear hole pattern	
■ Data Matrix code	
Turning cycles	
■ Area clearance cycles, longitudinal and transverse	Page 744
■ Recess turning cycles, radial and axial	
■ Recessing cycles, radial and axial	
■ Thread cutting cycles	
■ Simultaneous turning cycles	
■ Special cycles	

Cycle group	Further information
Special cycles	
■ Dwell time	Page 1209
■ Program call	Page 507
■ Tolerance	Page 971
■ Oriented spindle stop	Page 1231
■ Engraving	
■ Gear cycles	
■ Interpolation turning	
Grinding cycles	
■ Reciprocating stroke	Page 909
■ Dressing	
■ Compensation cycles	

Measuring cycles

Cycle group	Further information
Rotation	
■ Probing of plane, edge, two circles, beveled edge ■ Basic rotation ■ Two holes or studs ■ Via rotary axis ■ Via C-axis	Page 1600
Preset/Position	
■ Rectangle, inside or outside ■ Circle, inside or outside ■ Corner, inside or outside ■ Center of bolt circle, slot or ridge ■ Touch probe axis or single axis ■ Four holes	Page 1675
Measuring	
■ Angle ■ Circle, inside or outside ■ Rectangle, inside or outside ■ Slot or ridge ■ Bolt hole circle ■ Plane or coordinate	Page 1772
Special cycles	
■ Measuring or measuring in 3D ■ Probing in 3D ■ Fast probing	Page 1831
Calibrating the touch probe	
■ Calibrating the length ■ Calibration in a ring ■ Calibration on a stud ■ Calibration on a sphere	Page 1848
Measuring kinematics	
■ Saving the kinematics ■ Measure kinematics ■ Preset compensation ■ Kinematics grid	Page 1866
Measuring the tool (TT)	
■ Calibrating the TT ■ Tool length, radius or measuring completely ■ Calibrating the IR-TT ■ Lathe tool measurement	Page 1906

15.2 Technology-independent cycles

15.2.1 Overview

Cycle		Call	Further information
200	**DRILLING** ■ Basic hole ■ Input of the dwell time at top and bottom ■ Depth reference selectable	**CALL**-active	Page 488
201	**REAMING** ■ Reaming a hole ■ Input of the dwell time at bottom	**CALL**-active	Page 492
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	**CALL**-active	Page 494
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	**CALL**-active	Page 500

15.2.2 Cycle 200 DRILLING

ISO programming
G200

Application

With this cycle, you can drill basic holes. In this cycle, the depth reference is selectable.

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 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

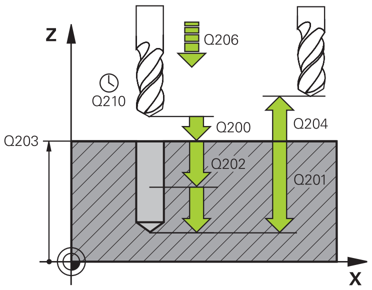
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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** or **PREDEF**
	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** or **PREDEF**
	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** or **PREDEF**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**

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  Y+20 FMAX M3
13 CYCL CALL
14 L  X+80  Y+50 FMAX M99
```

15.2.3 Cycle 201 REAMING

ISO programming
G201

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

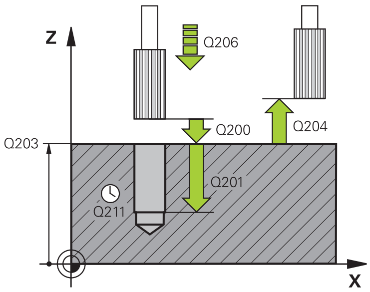
NOTICE
Danger of collision!
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!
▶ Enter depth as negative
▶ 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 can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- 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** or **PREDEF**
	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** or **PREDEF**
	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**, **PREDEF**
	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** or **PREDEF**

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  Y+20 FMAX M3
13 CYCL CALL
```

15.2.4 Cycle 203 UNIVERSAL DRILLING

ISO programming
G203

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.

Cycle sequence

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!

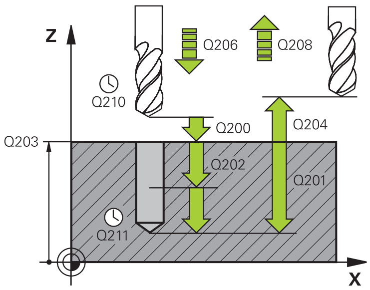
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- 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** or **PREDEF**
	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** or **PREDEF**
	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** or **PREDEF**
	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** or **PREDEF**
	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**, **PREDEF**
	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** or **PREDEF**
	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
```

15.2.5 Cycle 205 UNIVERSAL PECKING

ISO programming
G205

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 505

Cycle sequence

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 **Q373 FEED AFTER REMOVAL** 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

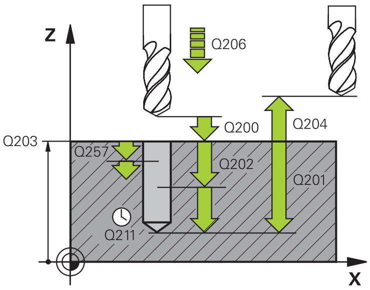
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

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** or **PREDEF**
	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** or **PREDEF**
	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** or **PREDEF**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	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**, **PREDEF**
	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**, **PREDEF**

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**
	Q373 Post-chip-removal approach feed? Traversing speed of the tool when approaching the advanced stop distance after chip removal. **0**: Move at **FMAX** **>0**: Feed in mm/min Input: **0...99999** or **FAUTO**, **FMAX**, **FU**, **FZ**

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 ~
    Q373=+0            ;FEED AFTER REMOVAL
```

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 ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 L X+30 Y+30 R0 FMAX M3	; Approach drilling position, spindle ON
7 CYCL CALL	; Cycle call
8 L Z+250 R0 FMAX	; Retract the tool, end program
9 M30	
10 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 **DEPTH Q201** 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 ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 L X+30 Y+30 R0 FMAX M3	; Approach drilling position, spindle ON
7 CYCL CALL	; Cycle call
8 L Z+250 R0 FMAX	; Retract the tool, end program
9 M30	
10 END PGM 205 MM	

15.3 Cycles for milling

15.3.1 Overview

Cycle		Call	Further information
202	**BORING** ■ Boring a hole ■ Input of the retraction feed rate ■ Input of the dwell time at bottom ■ Input of the retracting movement	CALL-active	Page 510
204	**BACK BORING** ■ Machining a counterbore on the underside of the workpiece ■ Input of the dwell time ■ Input of the retracting movement	CALL-active	Page 513
208	**BORE MILLING** ■ Milling of a hole ■ Input of a pre-drill diameter ■ Climb or up-cut milling selectable	CALL-active	Page 518
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	CALL-active	Page 521
240	**CENTERING** ■ Drilling a center hole ■ Input of the centering diameter or depth ■ Input of the dwell time at bottom	CALL-active	Page 531
206	**TAPPING** ■ With a floating tap holder ■ Input of the dwell time at bottom	CALL-active	Page 534
207	**RIGID TAPPING** ■ Without a floating tap holder ■ Input of the dwell time at bottom	CALL-active	Page 537
209	**TAPPING W/ CHIP BRKG** ■ Without a floating tap holder ■ Input of chip breaking behavior	CALL-active	Page 540
262	**THREAD MILLING** ■ Milling a thread into pre-drilled material	CALL-active	Page 546
263	**THREAD MLLNG/CNTSNKG** ■ Milling a thread into pre-drilled material ■ Machining a countersunk chamfer	CALL-active	Page 550
264	**THREAD DRILLNG/MLLNG**	CALL-active	Page 555

Cycle		Call	Further information
	■ Drilling into solid material ■ Milling a thread		
265	**HEL. THREAD DRLG/MLG** ■ Milling a thread into solid material	**CALL-** active	Page 560
267	**OUTSIDE THREAD MLLNG** ■ Milling an external thread ■ Machining a countersunk chamfer	**CALL-** active	Page 564
251	**RECTANGULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical	**CALL-** active	Page 569
252	**CIRCULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical or vertical	**CALL-** active	Page 575
253	**SLOT MILLING** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL-** active	Page 581
254	**CIRCULAR SLOT** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL-** active	Page 586
256	**RECTANGULAR STUD** ■ Roughing and finishing cycle ■ Approach position: selectable	**CALL-** active	Page 592
257	**CIRCULAR STUD** ■ Roughing and finishing cycle ■ Input of the start angle ■ Helical infeed starting from the workpiece blank diameter	**CALL-** active	Page 599
258	**POLYGON STUD** ■ Roughing and finishing cycle ■ Helical infeed starting from the workpiece blank diameter	**CALL-** active	Page 604
233	**FACE MILLING** ■ Roughing and finishing cycle ■ Roughing strategy and direction: selectable ■ Input of side walls	**CALL-** active	Page 609
20	**CONTOUR DATA** ■ Input of machining information	**DEF-** active	Page 623
21	**PILOT DRILLING** ■ Machining a hole for non-center cutting tools	**CALL-** active	Page 625
22	**ROUGH-OUT** ■ Roughing or fine roughing of the contour ■ Takes infeed points of the rough-out tool into account	**CALL-** active	Page 627

Cycle		Call	Further information
23	**FLOOR FINISHING** ■ Finishing with finishing allowance for the floor from Cycle **20**	**CALL**-active	Page 632
24	**SIDE FINISHING** ■ Finishing with side finishing allowance from Cycle **20**	**CALL**-active	Page 638
270	**CONTOUR TRAIN DATA** ■ Input of contour data for Cycle **25** or **276**	**DEF**-active	Page 638
25	**CONTOUR TRAIN** ■ Machining of open and closed contours ■ Monitoring for undercuts and contour damage	**CALL**-active	Page 640
275	**TROCHOIDAL SLOT** ■ Machining of open and closed contours using trochoidal milling.	**CALL**-active	Page 645
276	**THREE-D CONT. TRAIN** ■ Machining of open and closed contours ■ Detection of residual material ■ 3D contours—additional processing of coordinates from the tool axis	**CALL**-active	Page 651
271	**OCM CONTOUR DATA** (option 167) ■ Definition of the machining information for the contour or subprograms ■ Input of a bounding frame or block	**DEF**-active	Page 661
272	**OCM ROUGHING** (option 167) ■ Technology data for roughing contours ■ Use of the OCM cutting data calculator ■ Plunging behavior: vertical, helical, or reciprocating ■ Plunging strategy: selectable	**CALL**-active	Page 663
273	**OCM FINISHING FLOOR** (option 167) ■ Finishing with finishing allowance for the floor from Cycle **271** ■ Machining strategy with constant tool angle or with path calculated as equidistant (equal distances)	**CALL**-active	Page 678
274	**OCM FINISHING SIDE** (option 167) ■ Finishing with side finishing allowance from Cycle **271**	**CALL**-active	Page 681
277	**OCM CHAMFERING** (option 167) ■ Deburr the edges ■ Consider adjacent contours and walls	**CALL**-active	Page 683
291	**COUPLG.TURNG.INTERP.** (option 96) ■ Coupling of the tool spindle with the positions of the linear axes ■ Or, rescind the spindle coupling	**CALL**-active	Page 686

Cycle		Call	Further information
292	**CONTOUR.TURNG.INTRP.** (option 96) ■ Coupling of the tool spindle with the positions of the linear axes ■ Create certain rotationally symmetric contours in the active working plane ■ Possible with tilted working plane	**CALL**-active	Page 694
225	**ENGRAVING** ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc	**CALL**-active	Page 704
232	**FACE MILLING** ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan	**CALL**-active	Page 711
18	**THREAD CUTTING** ■ With controlled spindle ■ Spindle stops at the bottom of the hole	**CALL**-active	Page 718

15.3.2 Cycle 202 BORING

ISO programming
G202

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.

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 **MDI** application in the **Manual** 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!

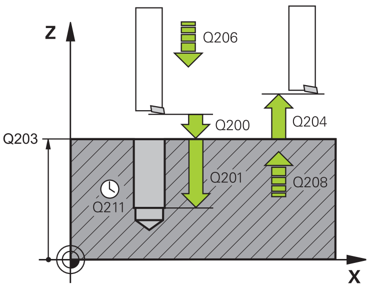
- Use **M137** to deactivate **M136** before the cycle start

- 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** or **PREDEF**
	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** or **PREDEF**
	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**, **PREDEF**
	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** or **PREDEF**
	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=+0                 ;DISENGAGING DIRECTN ~
    Q336=+0                 ;ANGLE OF SPINDLE ~
    Q357+0.2                ;CLEARANCE TO SIDE
13 L  X+30  Y+20 FMAX M3
14 CYCL CALL
15 L  X+80  Y+50 FMAX M99
```

15.3.3 Cycle 204 BACK BORING

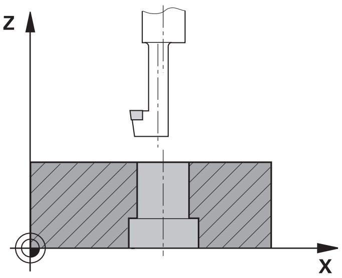
ISO programming
G204

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.

ℹ	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 **MDI** application in the **Manual** 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.

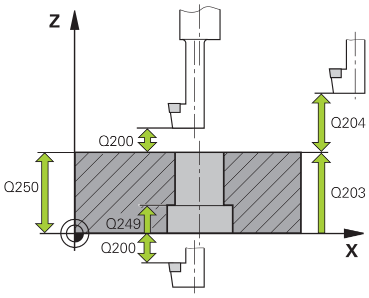
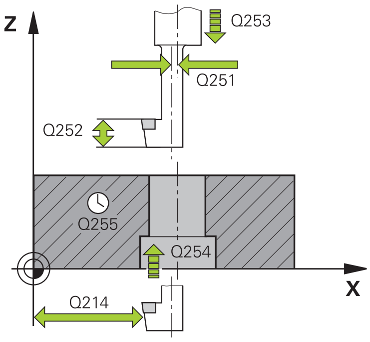
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- When calculating the starting point for boring, the control considers 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** or **PREDEF**
	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. Input:
	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**, **PREDEF**
	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** or **PREDEF**

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
```

15.3.4 Cycle 208 BORE MILLING

ISO programming
G208

Application

With this cycle, you can mill holes. In this cycle, you can define an optional, pre-drilled diameter. You can also program tolerances for the nominal diameter.

Cycle sequence

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 control moves on a semicircle for the first helical path while considering the path overlap **Q370**. The semicircle begins at the center of the hole.
3. The tool mills in a helix to the entered drilling depth at the programmed feed rate **F**.
4. When the drilling depth is reached, the control once again traverses a full circle to remove the material remaining after the initial plunge.
5. The control then centers the tool in the hole again and retracts it to set-up clearance **Q200**.
6. This procedure is repeated until the nominal diameter is reached (the control calculates the stepover by itself)
7. Finally, the tool is retracted to the set-up clearance or to the 2nd set-up clearance **Q204** 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**

> If you program **Q370=0** for the path overlap, the control uses the greatest path overlap possible for the first helical path. The control does this to prevent the tool from contacting the workpiece surface. All other paths are distributed uniformly.

Tolerances

The control allows you to store tolerances in the parameter **Q335 NOMINAL DIAMETER**.
You can define the following tolerances:

Tolerance	Example	Manufacturing dimension
Deviations	10+0.01-0.015	9.9975
DIN EN ISO 286-2	10H7	10.0075
ISO 2768-1	10m	10.0000

Proceed as follows:

- Start cycle definition
- Define cycle parameters
- Select the **TEXT** selection possibility in the action bar
- Enter a nominal dimension including tolerance

> - Machining is performed at mid-tolerance.
> - If you program an incorrect tolerance, the control interrupts machining with an error message.
> - Pay attention to capitalization when entering tolerances.

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

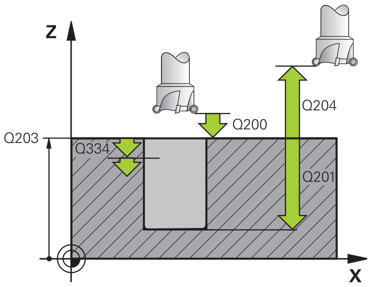
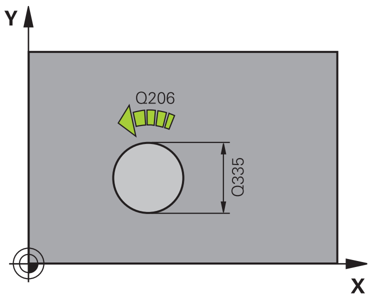
NOTICE
Caution: Danger to the workpiece and tool! If the selected infeed is too large, there is a danger of tool breakage and damage to the workpiece. ▶ Specify the maximum possible plunge angle and the corner radius **DR2** in the **ANGLE** column of the **TOOL.T** tool table. > The control automatically calculates the max. permissible infeed and changes your entered value accordingly, if necessary.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you have entered the bore hole diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation.
- An active mirror function **does not** influence the type of milling defined in the cycle.
- When calculating the overlap factor, the control takes the corner radius **DR2** of the current tool into account.
- 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.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues 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 lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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 during helical drilling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	**Q334 Feed per revolution of helix** Depth of the tool plunge with each helix (=360°). 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** or **PREDEF**
	Q335 Nominal diameter? Hole diameter. If you entered the nominal diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation. This value has an absolute effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 518 Input: **0...99999.9999**
	Q342 Roughing diameter? Enter the dimension of the pre-drilled diameter. This value has an absolute 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** or **PREDEF**
	Q370 Path overlap factor? The control uses the path overlap factor to determine the stepover factor k. **0**: The control uses the greatest path overlap possible for the first helical path. The control does this to prevent the tool from contacting the workpiece surface. All other paths are distributed uniformly. **>0**: The control multiplies the factor by the active tool radius. The result is the stepover factor k. Input: **0.1...1999** or **PREDEF**

Example

```
11 CYCL DEF 208 BORE MILLING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q201=-20                ;DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q334=+0.25              ;PLUNGING DEPTH ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q335=+5                 ;NOMINAL DIAMETER ~
    Q342=+0                 ;ROUGHING DIAMETER ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q370=+0                 ;TOOL PATH OVERLAP
12 CYCL CALL
```

15.3.5 Cycle 241 SINGLE-LIP D.H.DRLNG

ISO programming
G241

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.

Cycle sequence

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**
2 Depending on the positioning behavior, the control will either switch on the spindle with the programmed speed at the **SET-UP CLEARANCE Q200** or at a certain distance above the coordinate surface.

   Further information: "Position behavior when working with Q379", Page 527
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**:
   - If you have defined **Q435 DWELL DEPTH**, the control reduces the feed rate by **Q401 FEED RATE FACTOR** after the dwell depth has been reached and remains there for the duration of **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 **Q208 RETRACTION FEED RATE**.

   Further information: "Position behavior when working with Q379", Page 527
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!

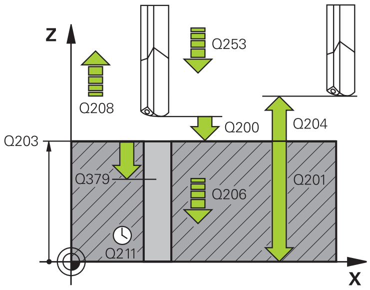
▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- 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** or **PREDEF**
	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** or **PREDEF**
	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** or **PREDEF**
	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**, **PREDEF**

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**, **PREDEF**
	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 526 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 the **DEPTH Q201**. **"..."**: 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 526 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

The 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 **PGM CALL** function, 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: "Variables: Q, QL, QR and QS parameters", Page 1362

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

15.3.6 Cycle 240 CENTERING

ISO programming

G240

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

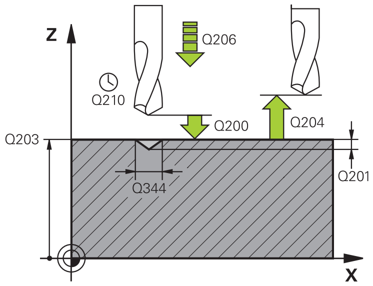
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

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** or **PREDEF**

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** or **PREDEF**

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** or **PREDEF**

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**, **PREDEF**

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  Y+20 R0 FMAX M3 M99
13 L  X+80  Y+50 R0 FMAX M99
```

15.3.7 Cycle 206 TAPPING

ISO programming
G206

Application

The thread is cut in one or more passes. A floating tap holder is used.

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 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

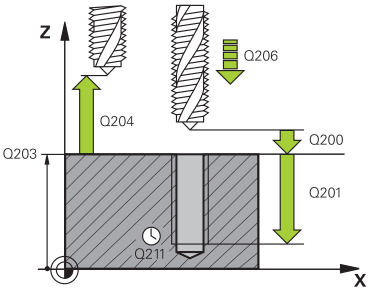
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

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** or **PREDEF**

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** or **PREDEF**

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** or **PREDEF**

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

Retracting in the Program Run, Single Block or Full Sequence mode

- To interrupt the program, select the **NC stop** key
- Select **MANUAL TRAVERSE**
- Retract the tool in the active tool axis
- To resume program execution, select **RESTORE POSITION**
- > A window is opened where the control shows the axis sequence as well as target position, current position and distance-to-go.
- Select the **NC start** key
- > The control moves the tool to the depth where it was stopped.
- To resume program execution, select **NC start** again

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

15.3.8 Cycle 207 RIGID TAPPING

ISO programming
G207

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.

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 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

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.

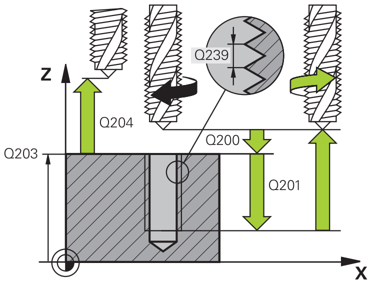
Notes on programming

- 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** or **PREDEF**
	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** or **PREDEF**

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 Program Run, Single Block or Full Sequence mode

- To interrupt the program, select the **NC stop** key
- Select **MANUAL TRAVERSE**
- Retract the tool in the active tool axis
- To resume program execution, select **RESTORE POSITION**
- > A window is opened where the control shows the axis sequence as well as target position, current position and distance-to-go.
- Select the **NC start** key
- > The control moves the tool to the depth where it was stopped.
- To resume program execution, select **NC start** again

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

15.3.9 Cycle 209 TAPPING W/ CHIP BRKG

ISO programming
G209

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 tool machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking.

Cycle sequence

1 The control positions the tool in the tool axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface.There, it carries out an oriented spindle stop
2 The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition. If you have defined a factor for increasing the spindle speed, the control retracts from the hole at the corresponding speed
3 It then reverses the direction of spindle rotation again and advances to the next infeed depth.
4 The control repeats this procedure (steps 2 to 3) until the programmed thread depth is reached
5 The tool is then retracted to set-up clearance. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
6 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. Synchronization may take place while the spindle is stationary.

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.
- 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.

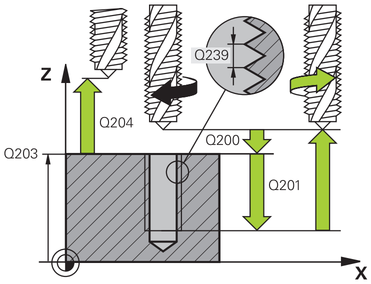
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.
- If you defined a speed factor for fast retraction in cycle parameter **Q403**, the control limits the speed to the maximum speed of the active gear stage.

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. Input: **0...99999.9999** or **PREDEF**
	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** or **PREDEF**
	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? The control multiplies the pitch **Q239** by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter **Q256** = 0, the control retracts the tool completely from the hole (to set-up clearance) for chip breaking. Input: **0...99999.9999**
	Q336 Angle for spindle orientation? Angle to which the control positions the tool before machining the thread. This allows you to re-cut the thread, if required. This value has an absolute effect. Input: **0...360**

Help graphic	Parameter
	Q403 RPM factor for retraction? Factor by which the control increases the spindle speed—and therefore also the retraction feed rate—when retracting from the drill hole. Maximum increase to maximum speed of the active gear stage. Input: **0.0001...10**

Example

```
11 CYCL DEF 209 TAPPING W/ CHIP BRKG ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q201=-18            ;DEPTH OF THREAD ~
    Q239=+1             ;THREAD PITCH ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q257=+0             ;DEPTH FOR CHIP BRKNG ~
    Q256=+1             ;DIST FOR CHIP BRKNG ~
    Q336=+0             ;ANGLE OF SPINDLE ~
    Q403=+1             ;RPM FACTOR
12 CYCL CALL
```

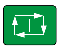
Retracting after a program interruption

Retracting in the Program Run, Single Block or Full Sequence mode

- To interrupt the program, select the **NC stop** key

- Select **MANUAL TRAVERSE**
- Retract the tool in the active tool axis

- To resume program execution, select **RESTORE POSITION**
- > A window is opened where the control shows the axis sequence as well as target position, current position and distance-to-go.



- Select the **NC start** key
- > The control moves the tool to the depth where it was stopped.
- To resume program execution, select **NC start** again

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

15.3.10 Fundamentals of thread milling

Requirements

- Your machine tool features internal spindle cooling (cooling lubricant at least 30 bars, compressed air supply at least 6 bars)
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer (you can set the compensation in **TOOL CALL** using the **DR** delta radius).
- If you are using a left-cutting tool (**M4**), the type of milling in **Q351** is reversed
- The working direction is determined by the following input parameters: Algebraic sign **Q239** (+ = right-hand thread / – = left-hand thread) and type of milling **Q351** (+1 = climb / –1 = up-cut).

  The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z+
Left-handed	–	–1(RR)	Z+
Right-handed	+	–1(RR)	Z–
Left-handed	–	+1(RL)	Z–

External thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z–
Left-handed	–	–1(RR)	Z–
Right-handed	+	–1(RR)	Z+
Left-handed	–	+1(RL)	Z+

NOTICE

Danger of collision!

If you program the plunging depth values with different algebraic signs a collision may occur.

- ▶ Make sure to program all depth values with the same algebraic sign. Example: If you program the **Q356** COUNTERSINKING DEPTH parameter with a negative sign, then **Q201** DEPTH OF THREAD must also have a negative sign
- ▶ If you want to repeat just the counterbore procedure in a cycle, you can enter 0 for DEPTH OF THREAD. In this case, the machining direction is determined by the programmed COUNTERSINKING DEPTH

NOTICE

Danger of collision!

A collision may occur if, upon tool breakage, you retract the tool from the hole in the direction of the tool axis only.

- ▶ Stop the program run if the tool breaks
- ▶ Switch to the **Manual operation** operating mode in the **MDI** application
- ▶ First move the tool in a linear movement towards the hole center
- ▶ Retract the tool in the tool axis direction

Programming and operating notes:

- The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle **8 MIRRORING** in only one axis.
- The programmed feed rate for thread milling references the cutting edge of the tool. However, since the control always displays the feed rate relative to the center path of the tool tip, the displayed value does not match the programmed value.

15.3.11 Cycle 262 THREAD MILLING

ISO programming
G262

Application

With this cycle, you can mill a thread into pre-drilled material.

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 moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3 The tool then approaches the nominal thread diameter tangentially in a helical movement. Before the helical approach, a compensating movement of the tool axis is carried out in order to begin at the programmed starting plane for the thread path
4 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
5 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
6 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the tool diameter is smaller than the nominal thread diameter by four times the thread pitch.

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! In the thread milling cycle, the tool will make a compensation movement in the tool axis before the approach. The length of the compensation movement is at most half of the thread pitch. This can result in a collision. ▶ Ensure sufficient space in the hole!

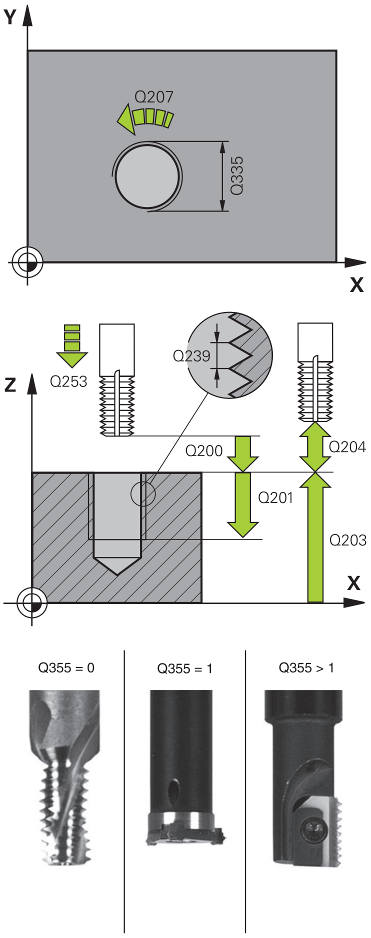
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you change the thread depth, the control will automatically move the starting point for the helical movement.

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 program the thread depth =0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter



Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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**

Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q355 Number of threads per step?

Number of thread revolutions by which the tool is moved:

0 = one helical line to the thread depth

1 = continuous helical path over the entire length of the thread

>1 = several helical paths with approach and departure; between them, the control offsets the tool by **Q355**, multiplied by the pitch.

Input: **0...99999**

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**, **PREDEF**

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** or **PREDEF**

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

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**

Help graphic	Parameter
	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** or **PREDEF**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 262 THREAD MILLING ~
    Q335=+5             ;NOMINAL DIAMETER ~
    Q239=+1             ;THREAD PITCH ~
    Q201=-18            ;DEPTH OF THREAD ~
    Q355=+0             ;THREADS PER STEP ~
    Q253=+750           ;F PRE-POSITIONING ~
    Q351=+1             ;CLIMB OR UP-CUT ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q207=+500           ;FEED RATE MILLING ~
    Q512=+0             ;FEED FOR APPROACH
12 CYCL CALL
```

15.3.12 Cycle 263 THREAD MLLNG/CNTSNKG

ISO programming

G263

Application

With this cycle, you can mill a thread into pre-drilled material. In addition, you can use it to machine a countersunk chamfer.

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

Countersinking

2 The tool moves at the feed rate for pre-positioning to the countersinking depth minus the set-up clearance, and then at the feed rate for countersinking to the countersinking depth.

3 If a set-up clearance to the side has been entered, the control immediately positions the tool at the pre-positioning feed rate to the countersinking depth.

4 Then, depending on the available space, the control smoothly approaches the tool to the core diameter, either tangentially from the center or with a pre-positioning movement to the side, and follows a circular path

Countersinking at front

5 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

6 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking

7 The tool then moves in a semicircle to the hole center

Thread milling

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

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

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

11 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

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

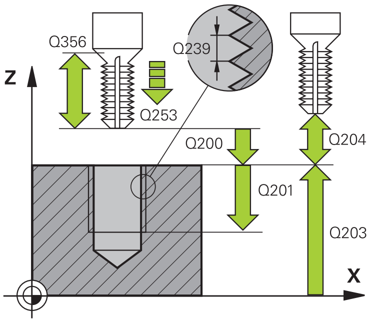
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Depth of thread
  2. Countersinking depth
  3. Depth at front

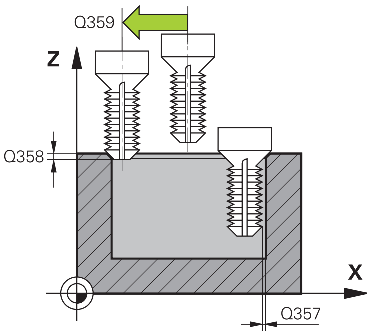
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.
- If you want to countersink at front, define the countersinking depth as 0.

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

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...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**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q356 Countersinking depth? Distance between tool point and the top surface of the workpiece. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	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**, **PREDEF**
	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** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	**Q357 Safety clearance to the side?** Distance between tool tooth and the wall. This value has an incremental effect. Input: **0...99999.9999**
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. 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** or **PREDEF**
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 263 THREAD MLLNG/CNTSNKG ~
    Q335=+5                 ;NOMINAL DIAMETER ~
    Q239=+1                 ;THREAD PITCH ~
    Q201=-18                ;DEPTH OF THREAD ~
    Q356=-20                ;COUNTERSINKING DEPTH ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q357=+0.2               ;CLEARANCE TO SIDE ~
    Q358=+0                 ;DEPTH AT FRONT ~
    Q359=+0                 ;OFFSET AT FRONT ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q254=+200               ;F COUNTERBORING ~
    Q207=+500               ;FEED RATE MILLING ~
    Q512=+0                 ;FEED FOR APPROACH
12 CYCL CALL
```

15.3.13 Cycle 264 THREAD DRILLNG/MLLNG

ISO programming
G264

Application

With this cycle, you can drill into solid material, machine a counterbore, and finally mill a thread.

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

Drilling

2 The tool drills to the first plunging depth at the programmed feed rate for plunging.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is retracted at rapid traverse to set-up clearance, and then moved again at **FMAX** to the entered advanced stop distance above the first plunging depth
4 The tool then advances with another infeed at the programmed feed rate.
5 The control repeats this procedure (steps 2 to 4) until the total drilling depth is reached

Countersinking at front

6 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
7 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking
8 The tool then moves in a semicircle to the hole center

Thread milling

9 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the algebraic sign of the thread pitch and the type of milling (climb or up-cut)
10 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion
11 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
12 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

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 algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Depth of thread
  2. Countersinking depth
  3. Depth at front

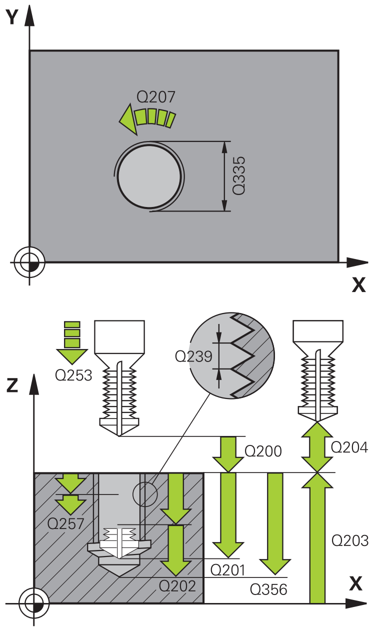
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

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

Cycle parameters

Help graphic | **Parameter**



Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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**

Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q356 Total hole depth?

Distance between workpiece surface and hole bottom. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

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**, **PREDEF**

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** or **PREDEF**

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.

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**

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**

Help graphic	Parameter
	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** or **PREDEF**
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. 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** or **PREDEF**
	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** or **PREDEF**
	Q206 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 264 THREAD DRILLNG/MLLNG ~
    Q335=+5                 ;NOMINAL DIAMETER ~
    Q239=+1                 ;THREAD PITCH ~
    Q201=-18                ;DEPTH OF THREAD ~
    Q356=-20                ;TOTAL HOLE DEPTH ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q258=+0.2               ;UPPER ADV STOP DIST ~
    Q257=+0                 ;DEPTH FOR CHIP BRKNG ~
    Q256=+0.2               ;DIST FOR CHIP BRKNG ~
    Q358=+0                 ;DEPTH AT FRONT ~
    Q359=+0                 ;OFFSET AT FRONT ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q207=+500               ;FEED RATE MILLING ~
    Q512=+0                 ;FEED FOR APPROACH
12 CYCL CALL
```

15.3.14 Cycle 265 HEL. THREAD DRLG/MLG

ISO programming
G265

Application

With this cycle, you can mill a thread into solid material. In addition, you can choose to machine a counterbore before or after milling the thread.

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

Countersinking at front

2 If countersinking occurs before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking occurs after thread milling, the control moves the tool to the countersinking depth at the feed rate for prepositioning
3 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking
4 The tool then moves in a semicircle to the hole center

Thread milling

5 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread
6 The tool then approaches the nominal thread diameter tangentially in a helical movement
7 The tool moves on a continuous helical downward path until the thread depth value is reached
8 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
9 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

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!

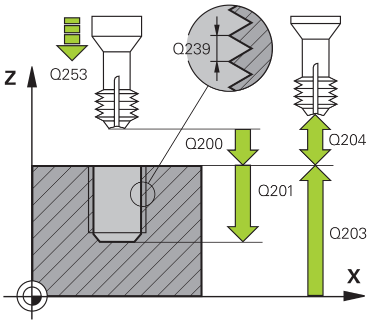
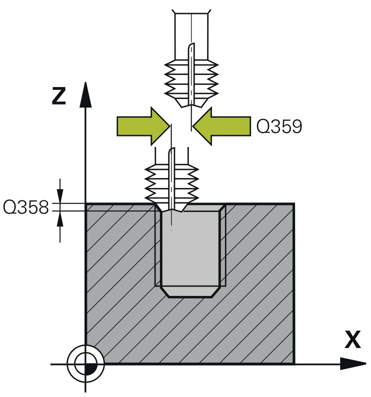
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you change the thread depth, the control will automatically move the starting point for the helical movement.
- The type of milling (up-cut or climb) is determined by the thread (right-hand or left-hand thread) and the direction of tool rotation, since it is only possible to work in the direction of the tool.
- The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Depth of thread
  2. Depth at front

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

Cycle parameters

Help graphic	Parameter
 	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...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**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	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**, **PREDEF**
 	**Q358 Sinking depth at front?** Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q360 Countersink (before/after:0/1)? Execution of the chamfer **0** = before thread machining **1** = after thread machining Input: **0**, **1**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 265 HEL. THREAD DRLG/MLG ~
    Q335=+5            ;NOMINAL DIAMETER ~
    Q239=+1            ;THREAD PITCH ~
    Q201=-18           ;DEPTH OF THREAD ~
    Q253=+750          ;F PRE-POSITIONING ~
    Q358=+0            ;DEPTH AT FRONT ~
    Q359=+0            ;OFFSET AT FRONT ~
    Q360=+0            ;COUNTERSINK PROCESS ~
    Q200=+2            ;SET-UP CLEARANCE ~
    Q203=+0            ;SURFACE COORDINATE ~
    Q204=+50           ;2ND SET-UP CLEARANCE ~
    Q254=+200          ;F COUNTERBORING ~
    Q207=+500          ;FEED RATE MILLING
12 CYCL CALL
```

15.3.15 Cycle 267 OUTSIDE THREAD MLLNG

ISO programming
G267

Application

With this cycle, you can mill an external thread. In addition, you can use it to machine a countersunk chamfer.

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

Countersinking at front

2 The control approaches the starting point for countersinking at front, starting from the center of the stud, on the reference axis in the working plane. The position of the starting point is determined by the thread radius, tool radius and pitch
3 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
4 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking
5 The tool then moves on a semicircle to the starting point

Thread milling

6 The control positions the tool at the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front
7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
8 The tool then approaches the nominal thread diameter tangentially in a helical movement
9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
10 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
11 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

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

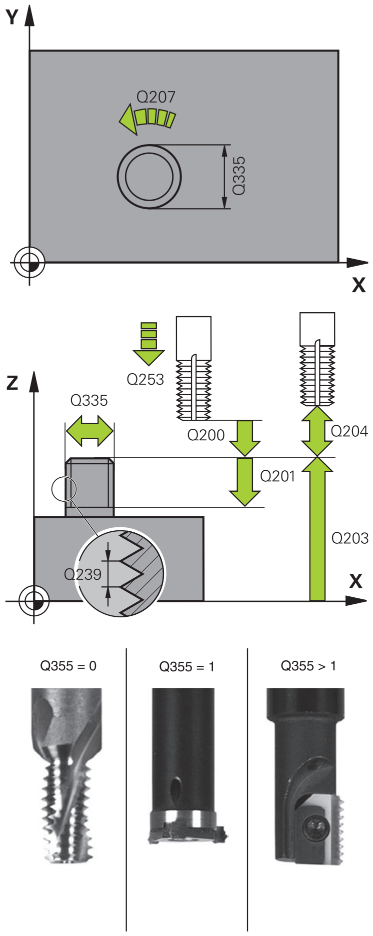
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).
- The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Depth of thread
  2. Depth at front

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

Cycle parameters

Help graphic



Parameter

Q335 Nominal diameter?

Nominal thread diameter

Input: **0...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**

Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q355 Number of threads per step?

Number of thread revolutions by which the tool is moved:

0 = one helical line to the thread depth

1 = continuous helical path over the entire length of the thread

>1 = several helical paths with approach and departure; between them, the control offsets the tool by **Q355**, multiplied by the pitch.

Input: **0...99999**

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**, **PREDEF**

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** or **PREDEF**

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. 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** or **PREDEF**
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
25 CYCL DEF 267  OUTSIDE THREAD MLLNG ~
    Q335=+10                ;NOMINAL DIAMETER ~
    Q239=+1.5               ;THREAD PITCH ~
    Q201=-20                ;DEPTH OF THREAD ~
    Q355=+0                 ;THREADS PER STEP ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q358=+0                 ;DEPTH AT FRONT ~
    Q359=+0                 ;OFFSET AT FRONT ~
    Q203=+30                ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q254=+150               ;F COUNTERBORING ~
    Q207=+500               ;FEED RATE MILLING ~
    Q512=+0                 ;FEED FOR APPROACH
```

15.3.16 Cycle 251 RECTANGULAR POCKET

ISO programming
G251

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. Specify the plunging strategy with parameter **Q366**.
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 tangentially 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 approach movement occurs on a radius in order to ensure a gentle approach. 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. The tool approaches the pocket floor tangentially

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

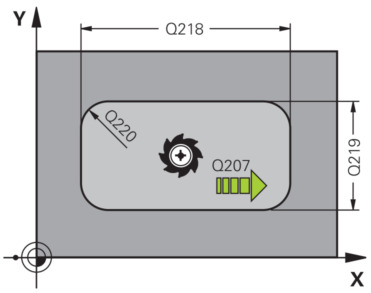
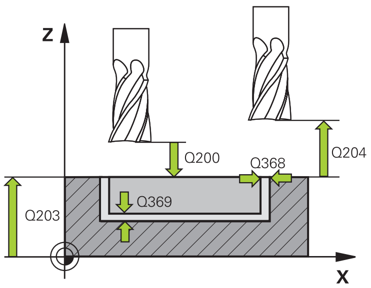
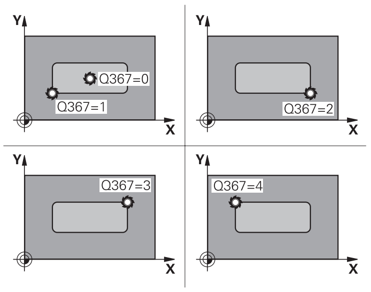
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed.
- 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.
- Cycle **251** takes the cutting width **RCUTS** from the tool table.

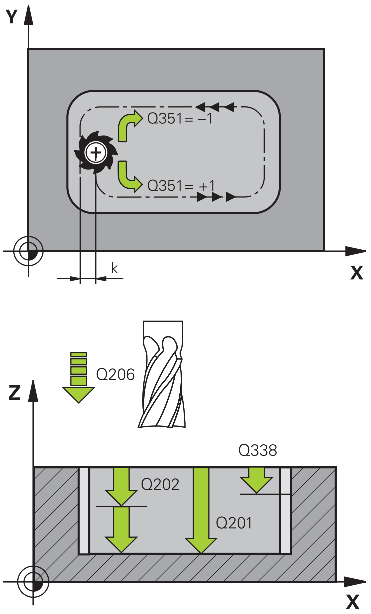
  Further information: "Plunging strategy Q366 with RCUTS", Page 575

Notes on programming

- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- 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**
	Q220 Corner radius? Radius of the pocket corner. If you have entered 0 here, the control assumes that the corner radius is equal to the tool radius. Input: **0...99999.9999**
	**Q368 Finishing allowance for side?** Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q224 Angle of rotation? Angle by which the entire operation is rotated. The center of rotation is the position at which the tool is located when the cycle is called. This value has an absolute effect. Input: **-360.000...+360.000**
	**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**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of pocket. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	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**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed 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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.41** or **PREDEF**
	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. If necessary, define the value of the **RCUTS** cutting width in the tool table **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. If necessary, define the value of the **RCUTS** cutting width in the tool table **PREDEF**: The control uses the value from the GLOBAL DEF block Input: **0**, **1**, **2** or **PREDEF** **Further information:** "Plunging strategy Q366 with RCUTS", Page 575
	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**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 251 RECTANGULAR POCKET ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q218=+60                ;FIRST SIDE LENGTH ~
    Q219=+20                ;2ND SIDE LENGTH ~
    Q220=+0                 ;CORNER RADIUS ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q224=+0                 ;ANGLE OF ROTATION ~
    Q367=+0                 ;POCKET POSITION ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q366=+1                 ;PLUNGE ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q439=+0                 ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

Plunging strategy Q366 with RCUTS

Helical plunging Q366 = 1

RCUTS > 0

- The control takes the cutting width **RCUTS** into account when calculating the helical path. The greater **RCUTS** is, the smaller the helical path.
- Formula for calculating the helical radius:

  $Helicalradius = R_{corr} - RCUTS$

  R_{corr}: Tool radius **R** + tool radius oversize **DR**
- If moving on a helical path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- The control does not monitor or modify the helical path.

Reciprocating plunge Q366 = 2

RCUTS > 0

- The control moves the tool along the complete reciprocating path.
- If moving on a reciprocating path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- The control moves the tool along one half of the reciprocating path.

15.3.17 Cycle 252 CIRCULAR POCKET

ISO programming
G252

Application

Use Cycle **252** to machine circular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

1. The control first moves the tool at rapid traverse to set-up clearance **Q200** above the workpiece
2. The tool plunges to the first plunging depth at the pocket center. Specify the plunging strategy with parameter **Q366**.
3. The control roughs out the pocket from the inside out, taking the path overlap (**Q370**) and the finishing allowances (**Q368** and **Q369**) into account.
4. At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall to set-up clearance **Q200** in the working plane, then retracts the tool by **Q200** at rapid traverse and returns it from there at rapid traverse to the pocket center
5. Steps 2 to 4 are repeated until the programmed pocket depth is reached, taking the finishing allowance **Q369** into account.
6. If only roughing was programmed (**Q215**=1), the tool moves away from the pocket wall tangentially by the set-up clearance **Q200**, then retracts at rapid traverse to the second set-up clearance **Q204** in the tool axis and returns at rapid traverse to the pocket center.

Finishing

1. If finishing allowances have been defined, the control first finishes the pocket walls, in multiple infeeds, if so specified.
2. The control positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200**
3. The control roughs out the pocket from the inside out, until the diameter **Q223** is reached
4. Then, the control again positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200** and repeats the finishing procedure for the side wall at the new depth
5. The control repeats this process until the programmed diameter is reached
6. After machining to the diameter **Q223**, the control retracts the tool tangentially by the finishing allowance **Q368** plus the set-up clearance **Q200** in the working plane, then retracts it at rapid traverse to set-up clearance **Q200** in the tool axis and returns it to the pocket center.
7. Next, the control moves the tool in the tool axis to the depth **Q201** and finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially.
8. The control repeats this process until the depth **Q201** plus **Q369** is reached.
9. Finally, the tool moves away from the pocket wall tangentially by the set-up clearance **Q200**, then retracts at rapid traverse to set-up clearance **Q200** in the tool axis and returns at rapid traverse to the pocket center.

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.
- 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.
- Cycle **252** takes the cutting width **RCUTS** from the tool table.

  Further information: "Plunging strategy Q366 with RCUTS", Page 581

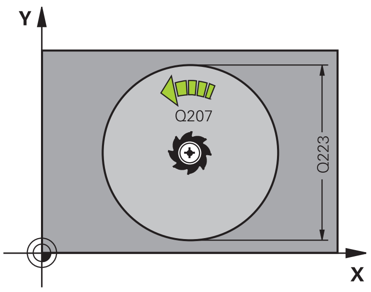
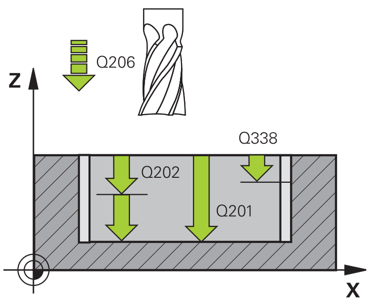
Notes on programming

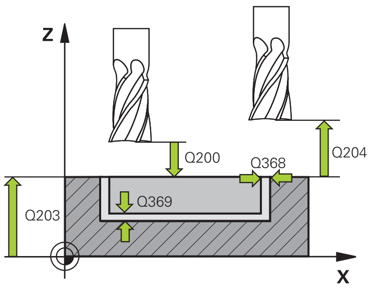
- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position (circle center) with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

Note regarding machine parameters

- For helical plunging, the control will display an error message if the internally calculated helix diameter is less than twice the tool diameter. If you are using a center-cut tool, you can switch this monitoring function off via the **suppressPlungeErr** machine parameter (no. 201006).

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**
	**Q223 Circle diameter?** Diameter of the finished pocket Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. 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**
	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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and bottom of pocket. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	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**

Help graphic	Parameter
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed 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** or **PREDEF**
	**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** or **PREDEF**
	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** or **PREDEF**
	Q366 Plunging strategy (0/1)? Type of plunging strategy: **0**: Vertical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as 0 or 90. Otherwise, the control will display an error message **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. If necessary, define the value of the **RCUTS** cutting width in the tool table Input: **0**, **1** or **PREDEF** **Further information:** "Plunging strategy Q366 with RCUTS", Page 581

Help graphic	Parameter
	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**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 252 CIRCULAR POCKET ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q223=+50                ;CIRCLE DIAMETER ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q366=+1                 ;PLUNGE ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q439=+0                 ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

Plunging strategy Q366 with RCUTS

Behavior with RCUTS

Helical plunging **Q366=1**:

RCUTS > 0

- The control takes the cutting width **RCUTS** into account when calculating the helical path. The greater **RCUTS** is, the smaller the helical path.
- Formula for calculating the helical radius:

  $Helicalradius = R_{corr} - RCUTS$

  R_{corr}: Tool radius **R** + tool radius oversize **DR**
- If moving on a helical path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- **suppressPlungeErr**=**on** (no. 201006)

  If moving on a helical path is not possible due to limited space, the control will reduce the helical path.
- **suppressPlungeErr**=**off** (no. 201006)

  If moving on a helical radius is not possible due to limited space, the control will display an error message.

15.3.18 Cycle 253 SLOT MILLING

ISO programming

G253

Application

Use Cycle **253** to completely machine a slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

1 Starting from the left slot arc center, the tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.

2 The control roughs out the slot from the inside out, taking the finishing allowances (**Q368** and **Q369**) into account

3 The control retracts the tool to set-up clearance **Q200**. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed

4 This process is repeated until the programmed slot depth is reached

Finishing

5 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

6 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!

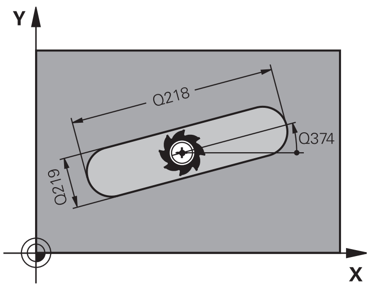
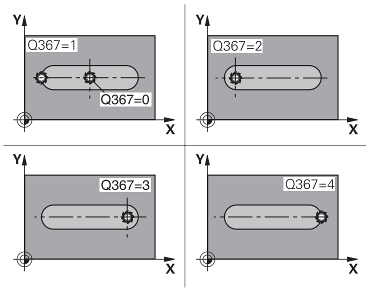
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

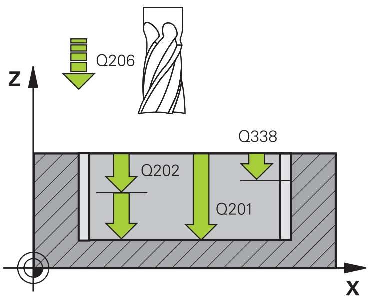
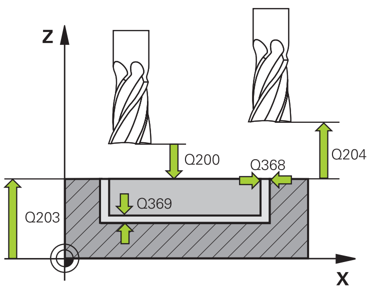
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.
- 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.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

Notes on programming

- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- 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)? 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 Length of slot?** Enter the length of the slot. It is parallel to the main axis of the working plane. 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. If the slot width equals the tool diameter, the control will mill an oblong hole. Maximum slot width for roughing: Twice the tool diameter Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q374 Angle of rotation? Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. This value has an absolute effect. Input: **-360.000...+360.000**
	**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**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	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**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed 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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0** = Vertical plunging. The plunging angle **ANGLE** in the tool table is not evaluated. **1, 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. Alternative: **PREDEF** Input: **0**, **1**, **2**
	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**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

11 CYCL DEF 253 SLOT MILLING ~	
Q215=+0	;MACHINING OPERATION ~
Q218=+60	;SLOT LENGTH ~
Q219=+10	;SLOT WIDTH ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q374=+0	;ANGLE OF ROTATION ~
Q367=+0	;SLOT POSITION ~
Q207=+500	;FEED RATE MILLING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q201=-20	;DEPTH ~
Q202=+5	;PLUNGING DEPTH ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q338=+0	;INFEED FOR FINISHING ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q366=+2	;PLUNGE ~
Q385=+500	;FINISHING FEED RATE ~
Q439=+3	;FEED RATE REFERENCE
12 L X+50 Y+50 R0 FMAX M99	

15.3.19 Cycle 254 CIRCULAR SLOT

ISO programming
G254

Application

Use Cycle **254** to completely machine a circular slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

1 The tool moves in a reciprocating motion in the slot center at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs out the slot from the inside out, taking the finishing allowances (**Q368** and **Q369**) into account
3 The control retracts the tool to set-up clearance **Q200**. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed
4 This process is repeated until the programmed slot depth is reached

Finishing

5 If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially
6 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

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.

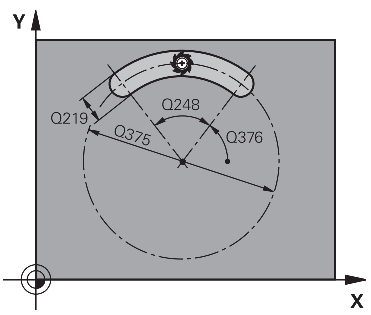
- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

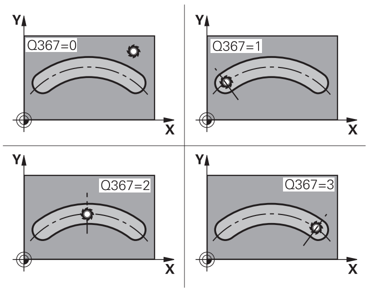
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.
- 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.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

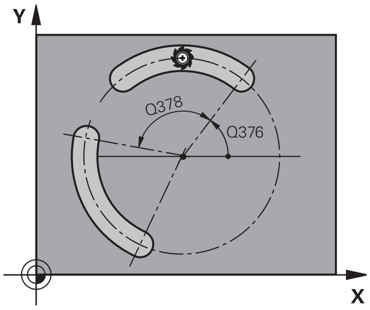
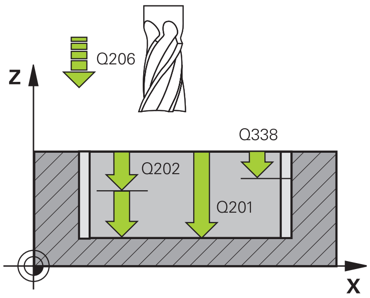
Notes on programming

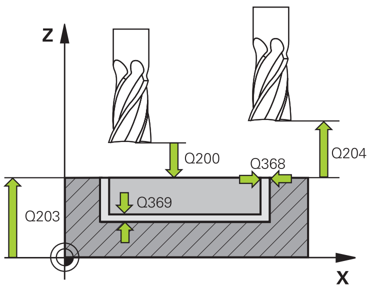
- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle 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.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.

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**
	**Q219 Width of slot?** Enter the width of the slot, which must be parallel to the secondary axis of the working plane. If the slot width equals the tool diameter, the control will mill an oblong hole. Maximum slot width for roughing: Twice the tool diameter Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q375 Pitch circle diameter? Enter the diameter of the circle. Input: **0...99999.9999**

Help graphic	Parameter
	**Q367 Ref. for slot pos. (0/1/2/3)?** Position of the slot relative to the position of the tool when the cycle is called: **0**: The tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle. **1**: Tool position = Center of left slot circle. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account. **2**: Tool position = Center of center line. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account. **3**: Tool position = Center of right slot circle. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account. Input: **0**, **1**, **2**, **3**
	Q216 Center in 1st axis? Center of the pitch circle in the main axis of the working plane. **Only effective if Q367 = 0**. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q217 Center in 2nd axis? Center of the pitch circle in the secondary axis of the working plane. **Only effective if Q367 = 0**. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	**Q376 Starting angle?** Enter the polar angle of the starting point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q248 Angular length? Enter the angular length of the slot. This value has an incremental effect. Input: **0...360**
	Q378 Intermediate stepping angle? Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. This value has an incremental effect. Input: **-360.000...+360.000**
	Q377 Number of repetitions? Number of machining operations on a pitch circle Input: **1...99999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	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**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed 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** or **PREDEF**
	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** or **PREDEF**
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0**: Vertical plunging. The plunging angle **ANGLE** in the tool table is not evaluated. **1, 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 **PREDEF**: The control uses the value from the GLOBAL DEF block. Input: **0**, **1**, **2**
	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
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 254 CIRCULAR SLOT ~
    Q215=+0             ;MACHINING OPERATION ~
    Q219=+10            ;SLOT WIDTH ~
    Q368=+0             ;ALLOWANCE FOR SIDE ~
    Q375=+60            ;PITCH CIRCLE DIAMETR ~
    Q367=+0             ;REF. SLOT POSITION ~
    Q216=+50            ;CENTER IN 1ST AXIS ~
    Q217=+50            ;CENTER IN 2ND AXIS ~
    Q376=+0             ;STARTING ANGLE ~
    Q248=+0             ;ANGULAR LENGTH ~
    Q378=+0             ;STEPPING ANGLE ~
    Q377=+1             ;NR OF REPETITIONS ~
    Q207=+500           ;FEED RATE MILLING ~
    Q351=+1             ;CLIMB OR UP-CUT ~
    Q201=-20            ;DEPTH ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q369=+0             ;ALLOWANCE FOR FLOOR ~
    Q206=+150           ;FEED RATE FOR PLNGNG ~
    Q338=+0             ;INFEED FOR FINISHING ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q366=+2             ;PLUNGE ~
    Q385=+500           ;FINISHING FEED RATE ~
    Q439=+0             ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

15.3.20 Cycle 256 RECTANGULAR STUD

ISO programming
G256

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) to the starting position for stud machining. Specify the starting position with parameter **Q437**. The default position (**Q437**=0) is 2 mm to the right of the stud blank
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 tangentially 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. If, on the other hand, you did not set the starting point on a side, but rather on a corner (**Q437** not equal to 0), the control mills on a spiral path from the starting point inward until the finished dimension has been reached.
5. If further stepovers are required, the tool is retracted from the contour on a tangential path 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
8. At the end of the cycle, the control positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position

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.

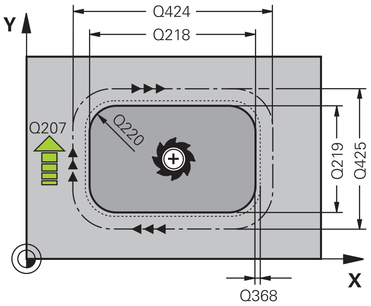
- Depending on the approach position **Q439**, leave enough room next to the stud for the approach movement
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position.

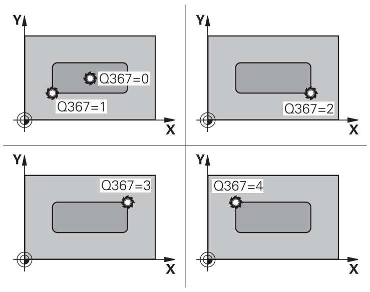
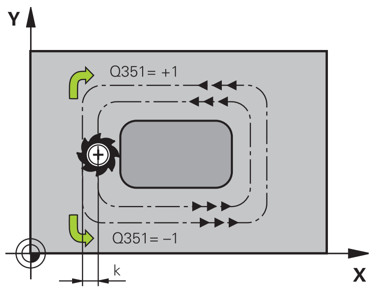
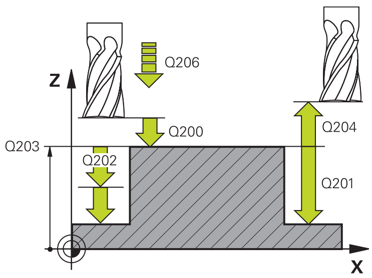
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the 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 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. 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. Input: **0...99999.9999**
	Q425 Workpiece blank side length 2? Length of stud blank parallel to the secondary axis of the working plane. Input: **0...99999.9999**
	Q220 Radius / Chamfer (+/-)? Enter the value for the radius or chamfer form element. If you enter a positive value, the control will round every corner. The value you enter here refers to the radius. If you enter a negative value, all corners of the contour will be chamfered with the value entered as the length of the chamfer. Input: **-99999.9999...+99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane, is left over after machining. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q224 Angle of rotation? Angle by which the entire operation is rotated. The center of rotation is the position at which the tool is located when the cycle is called. This value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**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**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	**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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and bottom of stud. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	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**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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**

Help graphic	Parameter
	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** or **PREDEF**
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.9999** or **PREDEF**
	Q437 Starting position (0...4)? Specify the approach strategy of the tool: **0**: From the right of the stud (default setting) **1**: Lower left corner **2**: Lower right corner **3**: Upper right corner **4**: Upper left corner If approach marks appear on the stud surface during approach with the setting **Q437**=0, then choose another approach position. Input: **0**, **1**, **2**, **3**, **4**
	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**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	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**

Example

```
11 CYCL DEF 256 RECTANGULAR STUD ~
    Q218=+60              ;FIRST SIDE LENGTH ~
    Q424=+75              ;WORKPC. BLANK SIDE 1 ~
    Q219=+20              ;2ND SIDE LENGTH ~
    Q425=+60              ;WORKPC. BLANK SIDE 2 ~
    Q220=+0               ;CORNER RADIUS ~
    Q368=+0               ;ALLOWANCE FOR SIDE ~
    Q224=+0               ;ANGLE OF ROTATION ~
    Q367=+0               ;STUD POSITION ~
    Q207=+500             ;FEED RATE MILLING ~
    Q351=+1               ;CLIMB OR UP-CUT ~
    Q201=-20              ;DEPTH ~
    Q202=+5               ;PLUNGING DEPTH ~
    Q206=+3000            ;FEED RATE FOR PLNGNG ~
    Q200=+2               ;SET-UP CLEARANCE ~
    Q203=+0               ;SURFACE COORDINATE ~
    Q204=+50              ;2ND SET-UP CLEARANCE ~
    Q370=+1               ;TOOL PATH OVERLAP ~
    Q437=+0               ;APPROACH POSITION ~
    Q215=+1               ;MACHINING OPERATION ~
    Q369=+0               ;ALLOWANCE FOR FLOOR ~
    Q338=+0               ;INFEED FOR FINISHING ~
    Q385=+500             ;FEED RATE FOR FINISHING
12 L  X+50  Y+50 R0 FMAX M99
```

15.3.21 Cycle 257 CIRCULAR STUD

ISO programming
G257

Application

Use Cycle **257** to machine a circular stud. The control mills the circular stud with a helical infeed motion starting from the workpiece blank diameter.

Cycle sequence

1 If the current position of the tool is below the 2nd set-up clearance, the control then lifts it off and retracts it to the 2nd set-up clearance.
2 The tool moves from the stud center to the starting position for stud machining. With the polar angle, you specify the starting position with respect to the stud center using parameter **Q376**.
3 The control moves the tool at rapid traverse **FMAX** to set-up clearance **Q200**, and from there advances to the first plunging depth at the feed rate for plunging
4 The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5 The control retracts the tool from the contour by 2 mm on a tangential path
6 If more than one plunging movement is required, the tool repeats the plunging movement at the point next to the departure movement
7 This process is repeated until the programmed stud depth is reached
8 At the end of the cycle, the tool firsts departs on a tangential path and is then retracted in the tool axis to the 2nd set-up clearance defined in the cycle. This means that the end position differs from the starting position

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 danger of collision if there is insufficient room next to the stud. ▶ Check the machining sequence using the graphic simulation.

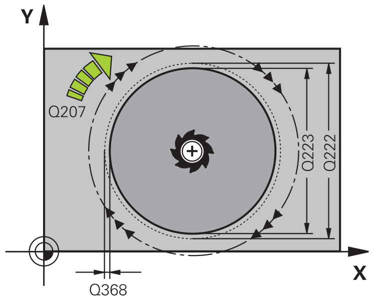
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- 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 (stud center) 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**



Q223 Finished part diameter?

Diameter of the finished stud

Input: **0...99999.9999**

Q222 Workpiece blank diameter?

Diameter of workpiece blank. The workpiece blank diameter must be greater than the diameter of the finished part. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover.

Input: **0...99999.9999**

Q368 Finishing allowance for side?

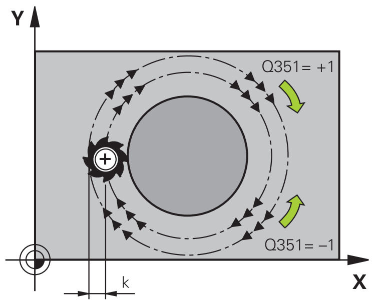
Finishing allowance in the working plane. This value has an incremental effect.

Input: **-99999.9999...+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**



Q351 Direction? Climb=+1, Up-cut=-1

Type of milling operation. The direction of spindle rotation is taken into account.

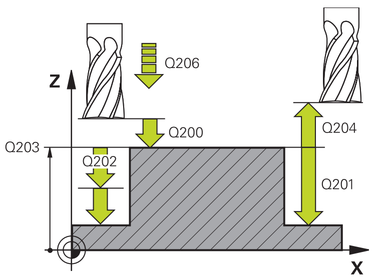
+1 = climb milling

-1 = up-cut milling

PREDEF: The control uses the value of a **GLOBAL DEF** block

(If you enter 0, climb milling is performed)

Input: **-1**, **0**, **+1** or **PREDEF**



Q201 Depth?

Distance between workpiece surface and bottom of stud. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q202 Plunging depth?

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

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**

Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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** or **PREDEF**
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.9999** or **PREDEF**
	Q376 Starting angle? Polar angle relative to the stud center, from which the tool approaches the stud. Input: **-1...+359**
	Q215 Machining operation (0/1/2)? Specify the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect.
	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**

Example

```
11 CYCL DEF 257 CIRCULAR STUD ~
    Q223=+50                ;FINISHED PART DIA. ~
    Q222=+52                ;WORKPIECE BLANK DIA. ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q206=+3000              ;FEED RATE FOR PLNGNG ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q376=-1                 ;STARTING ANGLE ~
    Q215=+1                 ;MACHINING OPERATION ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q385=+500               ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

15.3.22 Cycle 258 POLYGON STUD

ISO programming
G258

Application

Use Cycle **258** to machine a regular polygon by machining the contour outside. The milling operation is carried out on a spiral path based on the diameter of the workpiece blank.

Cycle sequence

1 If, at the beginning of machining, the work piece is positioned below the 2nd set-up clearance, the control will retract the tool back to 2nd set-up clearance
2 Starting from the center of the stud the control moves the tool to the starting point of stud machining. The starting point depends, among other things, on the diameter of the workpiece blank and the angle of rotation of the stud. The angle of rotation is determined with parameter **Q224**
3 The tool moves at rapid traverse **FMAX** to the setup clearance **Q200** and from there with the feed rate for plunging to the first plunging depth.
4 The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5 The control moves the tool on a tangential path from the outside to the inside
6 The tool will be lifted in the direction of the spindle axis to 2nd set-up clearance in one rapid movement
7 If several plunging depths are required, the control returns the tool to the starting point of the stud milling process and then plunges the tool to the programmed depth
8 This process is repeated until the programmed stud depth is reached.
9 At the end of the cycle, first a departing motion is performed. Then the control will move the tool on the tool axis to 2nd set-up clearance

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!

In this cycle, the control performs an automatic approach movement. If there is not enough space, a collision might occur.

- Use **Q224** to specify which angle is used to machine the first corner of the polygon stud. Input range: –360° to +360°
- Depending on the angle of rotation **Q224**, the following amount of space must be left next to the stud: At least tool diameter +2 mm

NOTICE

Danger of collision!

At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle need not be the same as the starting position. There is a danger of collision!

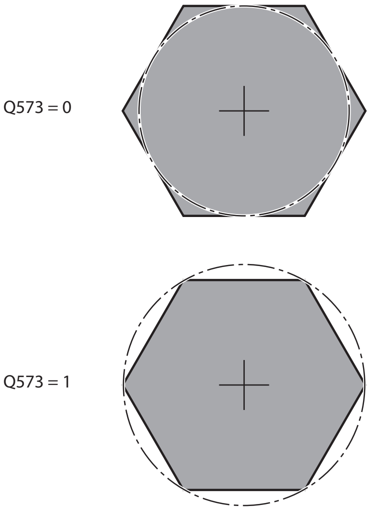
- Control the traversing movements of the machine
- In the **Simulation** workspace of the **Editor** operating mode, check the end position of the tool after the cycle
- After the cycle, program the absolute (not incremental) coordinates

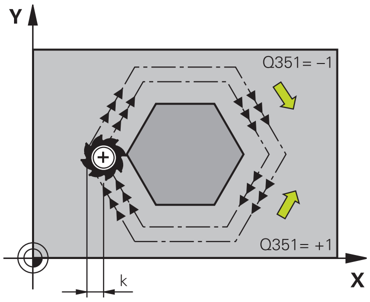
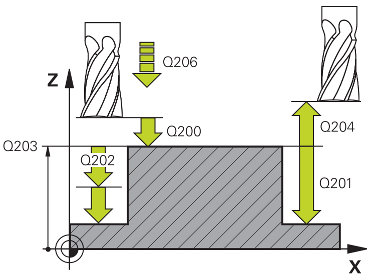
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- 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

- Before the start of the cycle you will have to pre-position the tool in the working plane. In order to do so, move the tool with radius compensation **R0** to the center of the stud.
- 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
	**Q573 Inscr.circle/circumcircle (0/1)?** Define whether the dimension **Q571** is referenced to the inscribed circle or the circumcircle: **0**: Dimension is referenced to the inscribed circle **1**: Dimension is referenced to the circumcircle Input: **0**, **1**
	Q571 Reference circle diameter? Enter the diameter of the reference circle. Specify in parameter **Q573** whether the diameter entered here is referenced to the inscribed circle or the circumcircle. You can program a tolerance if needed. Input: **0...99999.9999**
	Q222 Workpiece blank diameter? Enter the diameter of the blank. The workpiece blank diameter must be greater than the reference circle diameter. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input: **0...99999.9999**
	Q572 Number of corners? Enter the number of corners of the polygon stud. The control distributes the corners evenly on the stud. Input: **3...30**
	Q224 Angle of rotation? Specify which angle is used to machine the first corner of the polygon stud. Input: **-360.000...+360.000**
	Q220 Radius / Chamfer (+/-)? Enter the value for the radius or chamfer form element. If you enter a positive value, the control will round every corner. The value you enter here refers to the radius. If you enter a negative value, all corners of the contour will be chamfered with the value entered as the length of the chamfer. Input: **-99999.9999...+99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. If you enter a negative value here, the control will return the tool to a diameter outside of the workpiece blank diameter after roughing. This value has an incremental effect. Input: **-99999.9999...+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**

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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of stud. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	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**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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** or **PREDEF**
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.9999** or **PREDEF**

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**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	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**

Example

```
11 CYCL DEF 258 POLYGON STUD ~
    Q573=+0             ;REFERENCE CIRCLE ~
    Q571=+50            ;REF-CIRCLE DIAMETER ~
    Q222=+52            ;WORKPIECE BLANK DIA. ~
    Q572=+6             ;NUMBER OF CORNERS ~
    Q224=+0             ;ANGLE OF ROTATION ~
    Q220=+0             ;RADIUS / CHAMFER ~
    Q368=+0             ;ALLOWANCE FOR SIDE ~
    Q207=+500           ;FEED RATE MILLING ~
    Q351=+1             ;CLIMB OR UP-CUT ~
    Q201=-20            ;DEPTH ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q206=+3000          ;FEED RATE FOR PLNGNG ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q370=+1             ;TOOL PATH OVERLAP ~
    Q215=+0             ;MACHINING OPERATION ~
    Q369=+0             ;ALLOWANCE FOR FLOOR ~
    Q338=+0             ;INFEED FOR FINISHING ~
    Q385=+500           ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

15.3.23 Cycle 233 FACE MILLING

ISO programming
G233

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:

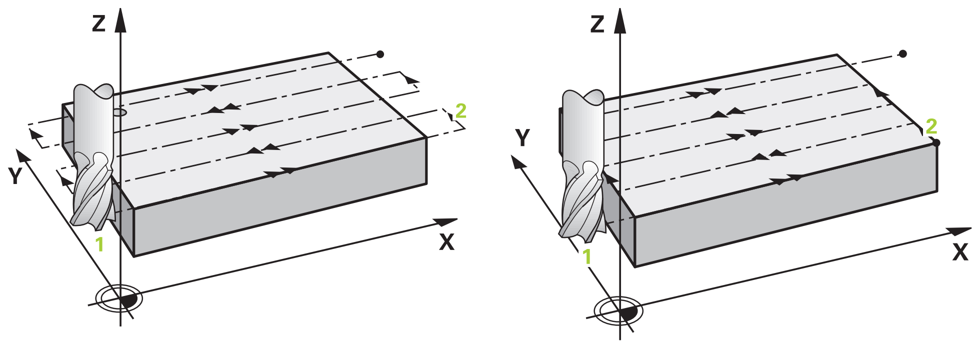
- **Strategy Q389=0**: Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover when retracting at rapid traverse
- **Strategy Q389=3**: The surface is machined line by line without overtravel; stepover when retracting at rapid traverse
- **Strategy Q389=4**: Helical machining from the outside toward the inside

Related topics

- Cycle **232 FACE MILLING**

  Further information: "Cycle 232 FACE MILLING ", Page 711

Strategies Q389=0 and Q389 =1

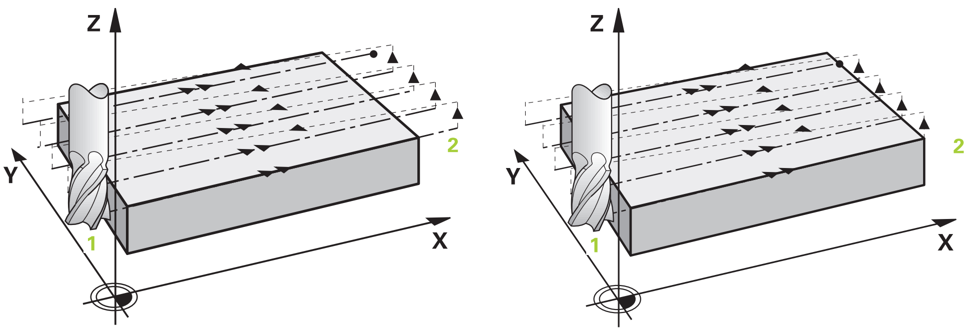


The strategies **Q389**=0 and **Q389**=1 differ in the overtravel during face milling. If **Q389**=0, the end point lies outside of the surface, with **Q389**=1, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the 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



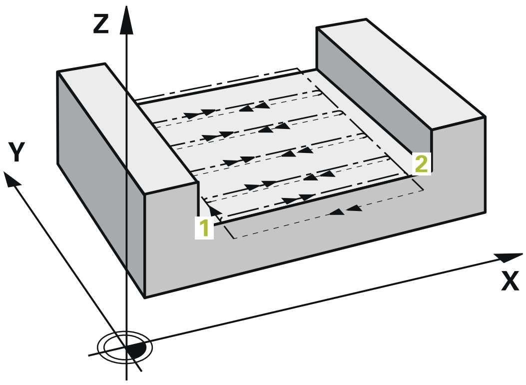
The strategies **Q389**=2 and **Q389**=3 differ in the overtravel during face milling. If **Q389**=2, the end point lies outside of the surface, with **Q389**=3, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the 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** directly 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**.

Strategies Q389=2 and Q389=3—with lateral limitation

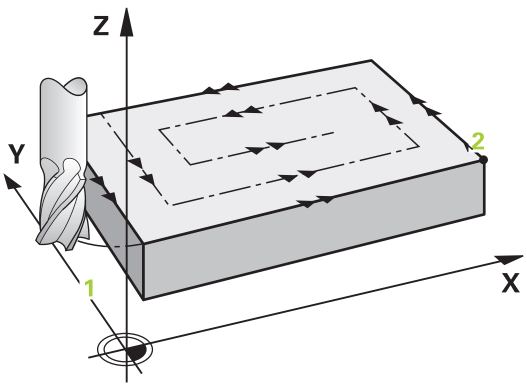
If you program a lateral limitation, the control might not be able to perform movements outside of the contour. In this case the cycle runs as follows:



1 The control positions the tool at **FMAX** to the starting point in the working plane. This position is offset from the edge of the workpiece by the tool radius and the set-up clearance **Q357** to the side.
2 The tool moves at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** and from there at **Q207 FEED RATE MILLING** to the first plunging depth **Q202**.
3 The control moves the tool on a circular path to the starting point **1**.
4 The tool moves at the programmed feed rate **Q207** to the end point **2** and departs from the contour on a circular path.
5 Then the control moves the tool to the approach position of the next path at **Q253 F PRE-POSITIONING** .
6 Steps 3 to 5 are repeated until the entire surface is milled.
7 If more than one infeed depth is programmed, the control moves the tool at the end of the last path to the set-up clearance **Q200** and positions in the working plane to the next approach position.
8 In the last infeed the control mills the **Q369 ALLOWANCE FOR FLOOR** at **Q385 FINISHING FEED RATE**.
9 At the end of the last path, the control retracts the tool to the 2nd set-up clearance **Q204** and then to the position last programmed before the cycle.

- The circular paths for approaching and departing the paths depend on **Q220 CORNER RADIUS**.
- 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**.

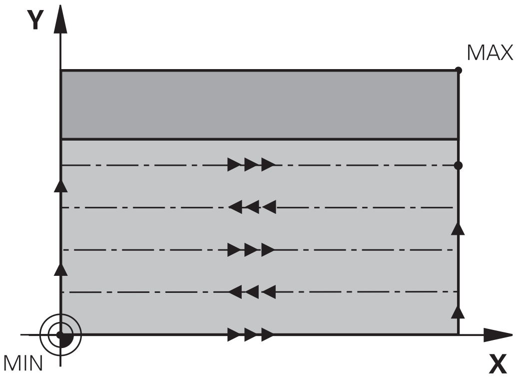
Strategy Q389=4



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 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.

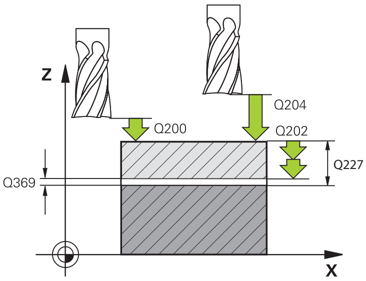
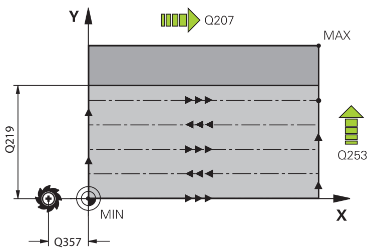
Notes on programming

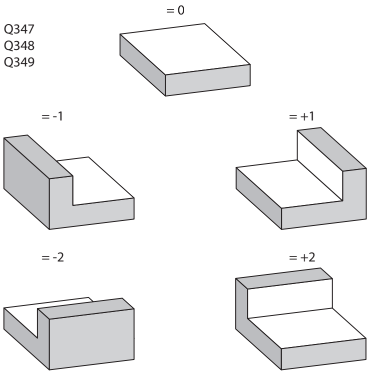
- Pre-position the tool in the working plane to the starting position with radius compensation R0. Note the machining direction.
- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- If 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? Value used for the last infeed. 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**, **PREDEF**

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** or **PREDEF**
	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** or **PREDEF**
	**Q347 1st limit?** Select the side of the workpiece where the plane surface is bordered by a side wall (not possible with helical machining). 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**
	Q220 Corner radius? Radius of a corner at limits (**Q347** to **Q349**) Input: **0...99999.9999**

Help graphic	Parameter
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. 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 ~
    Q220=+0                   ;CORNER RADIUS ~
    Q368=+0                   ;ALLOWANCE FOR SIDE ~
    Q338=+0                   ;INFEED FOR FINISHING ~
    Q367=-1                   ;SURFACE POSITION
12 L  X+50  Y+50 R0 FMAX M99
```

15.3.24 SL cycles

General information

SL Cycles enable you to form complex contours by combining up to twelve subcontours (pockets or islands). You define the individual subcontours in subprograms. The control calculates the entire contour from the list of subcontours (subprogram numbers) you have specified in Cycle **14 CONTOUR**.

> Programming and operating notes:
> - The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
> - SL Cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always use the simulation to verify your program before running it. This is a simple way of finding out whether the program calculated by the control will provide the desired results.
> - If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Characteristics of the subprograms

- Closed contour without approach and departure movements
- Coordinate transformations are permitted; if they are programmed within the subcontours, they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The control recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR
- The control recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL
- The subprograms must not contain spindle axis coordinates.
- Always program both axes in the first NC block of the subprogram
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms
- Without machining cycles, feed rates, and M functions

Cycle properties

- The control automatically positions the tool to the set-up clearance before each cycle. You must move the tool to a safe position before the cycle call
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling

The machining data, such as milling depth, allowances, and set-up clearance can be entered centrally in Cycle **20 CONTOUR DATA**.

Program structure: Machining with SL Cycles

```
0 BEGIN SL 2 MM
...
12 CYCL DEF 14 CONTOUR
...
13 CYCL DEF 20 CONTOUR DATA
...
16 CYCL DEF 21 PILOT DRILLING
...
17 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING
...
23 CYCL CALL
...
26 CYCL DEF 24 SIDE FINISHING
...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL2 MM
```

15.3.25 Cycle 20 CONTOUR DATA

ISO programming
G120

Application

Use Cycle **20** to specify machining data for the subprograms describing the subcontours.

Related topics

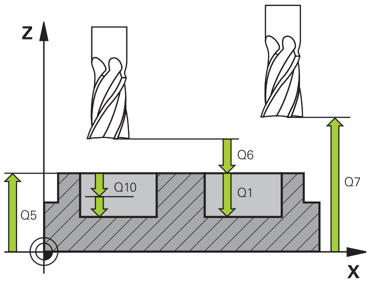
- Cycle **271 OCM CONTOUR DATA** (option 167)
  Further information: "Cycle 271 OCM CONTOUR DATA (option 167) ", Page 661

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **20** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **20** are valid for Cycles **21** to **24**.
- If you are using the SL cycles in **Q** parameter programs, the cycle parameters **Q1** to **Q20** cannot be used as program parameters.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH = 0, the control performs the cycle at the depth 0.

Cycle parameters

Help graphic	Parameter



Q1 Milling depth?

Distance between workpiece surface and pocket floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q2 Path overlap factor?

Q2 x tool radius = stepover factor k

Input: **0.0001...1.9999**

Q3 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q4 Finishing allowance for floor?

Finishing allowance for the floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q5 Workpiece surface coordinate?

Absolute coordinate of the top surface of the workpiece

Input: **-99999.9999...+99999.9999**

Q6 Set-up clearance?

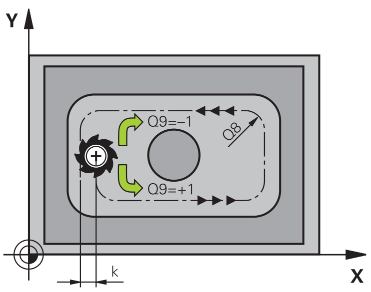
Distance between tool tip and the top surface of the workpiece. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q7 Clearance height?

Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). This value has an absolute effect.

Input: **-99999.9999...+99999.9999**



Q8 Inside corner radius?:

Inside "corner" rounding radius; entered value is referenced to the path of the tool center and is used to calculate smoother traverse motions between the contour elements.

Q8 is not a radius that is inserted between programmed elements as a separate contour element.

Input: **0...99999.9999**

Q9 Direction of rotation? cw = -1

Machining direction for pockets

Q9 = –1 up-cut milling for pocket and island

Q9 = +1 climb milling for pocket and island

Input: **-1**, **0**, **+1**

Example

11 CYCL DEF 20 CONTOUR DATA ~	
Q1=-20	;MILLING DEPTH ~
Q2=+1	;TOOL PATH OVERLAP ~
Q3=+0.2	;ALLOWANCE FOR SIDE ~
Q4=+0.1	;ALLOWANCE FOR FLOOR ~
Q5=+0	;SURFACE COORDINATE ~
Q6=+2	;SET-UP CLEARANCE ~
Q7=+50	;CLEARANCE HEIGHT ~
Q8=+0	;ROUNDING RADIUS ~
Q9=+1	;ROTATIONAL DIRECTION

15.3.26 Cycle 21 PILOT DRILLING

ISO programming
G121

Application

Use Cycle **21 PILOT DRILLING** if you machine a contour and then use a tool for roughing it out which has no center-cut end mill (ISO 1641). This cycle drills a hole in the area that will be roughed out later with a cycle such as Cycle **22**. Cycle **21** takes the finishing allowance for side and the finishing allowance for floor as well as the radius of the rough-out tool into account for the cutter infeed points. The cutter infeed points also serve as starting points for roughing.

Before programming the call of Cycle **21** you need to program two further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**—required by Cycle **21 PILOT DRILLING** to determine the drilling position in the plane
- Cycle **20 CONTOUR DATA**—required by Cycle **21 PILOT DRILLING** to determine parameters such as the hole depth and the set-up clearance

Cycle sequence

1. The control first positions the tool in the plane (the position results from the contour that you previously defined with Cycle **14** or **SEL CONTOUR**, and from the information on the rough-out tool)
2. The tool then moves at rapid traverse **FMAX** to set-up clearance. (specify the set-up clearance in Cycle **20 CONTOUR DATA**)
3. The tool drills from the current position to the first plunging depth at the programmed feed rate **F**.
4. Then, the tool retracts at rapid traverse **FMAX** to the starting position and advances again to the first plunging depth minus the advanced stop distance t
5. The advanced stop distance is automatically calculated by the control:
   - At a total hole depth up to 30 mm: t = 0.6 mm
   - At a total hole depth exceeding 30 mm: t = hole depth / 50
   - Maximum advanced stop distance: 7 mm
6. The tool then advances with another infeed at the programmed feed rate **F**.
7. The control repeats this procedure (steps 1 to 4) until the total hole depth is reached. The finishing allowance for floor is taken into account
8. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

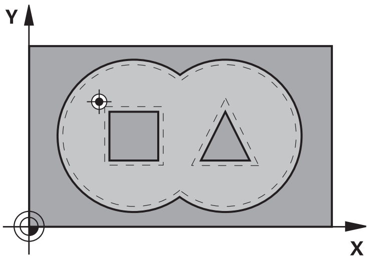
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- When calculating the infeed points, the control does not account for the delta value **DR** programmed in a **TOOL CALL** block.
- In narrow areas, the control may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.
- If **Q13**=0, the control uses the data of the tool that is currently in the spindle.

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining. After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have programmed **ToolAxClearanceHeight**.

Cycle parameters

Help graphic	Parameter
	**Q10 Plunging depth?** Tool infeed per cut (minus sign for negative machining direction). This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q13 or **QS13 Rough-out tool number/name?** Number or name of the rough-out tool. You are able to transfer the tool directly from the tool table via the selection option in the action bar. Input: **0...999999.9** or max. **255** characters

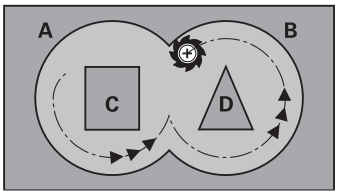
Example

```
11 CYCL DEF 21 PILOT DRILLING ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q13=+0                  ;ROUGH-OUT TOOL
```

15.3.27 Cycle 22 ROUGH-OUT

ISO programming
G122

Application



Use Cycle **22 ROUGH-OUT** to define the technology data for roughing.

Before programming the call of Cycle **22**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable

Related topics

- Cycle **272 OCM ROUGHING** (option 167)
  Further information: "Cycle 272 OCM ROUGHING (option 167) ", Page 663

Cycle sequence

1 The control positions the tool above the cutter infeed point, taking the finishing allowance for side into account
2 After reaching the first plunging depth, the tool mills the contour in an outward direction at the programmed milling feed rate **Q12**
3 The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B)
4 In the next step, the control moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached
5 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane. There is a danger of collision!

- After the end of the cycle, position the tool with all coordinates of the working plane (e.g., **L X+80 Y+0 R0 FMAX**)
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- During fine roughing, the control does not take a defined wear value **DR** of the coarse roughing tool into account.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q1**, the control will display an error message.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

This cycle might require a center-cut end mill (ISO 1641) or pilot drilling with Cycle **21**.

Notes on programming

- If you clear out an acute inside corner and use an overlap factor greater than 1, some material might be left over. Check especially the innermost path in the test run graphic and, if necessary, change the overlap factor slightly. This allows another distribution of cuts, which often provides the desired results.
- Define the plunging behavior of Cycle **22** with parameter **Q19** and in the **ANGLE** and **LCUTS** columns of the tool table:
  - If **Q19** = 0 is defined, the tool will always plunge perpendicularly, even if a plunge angle (**ANGLE**) was defined for the active tool
  - If you define **ANGLE** = 90°, the control will plunge perpendicularly. The reciprocation feed rate **Q19** is used as plunging feed rate
  - If the reciprocation feed rate **Q19** is defined in Cycle **22** and **ANGLE** is between 0.1 and 89.999 in the tool table, the tool plunges helically using the defined **ANGLE**
  - If the reciprocation feed is defined in Cycle **22** and no **ANGLE** is defined in the tool table, the control displays an error message
  - If the geometry conditions do not allow helical plunging (slot geometry), the control tries a reciprocating plunge (the reciprocation length is calculated from **LCUTS** and **ANGLE** (reciprocation length = **LCUTS** / tan **ANGLE**))

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q18 or **QS18 Coarse roughing tool?** Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters
	Q19 Feed rate for reciprocation? Reciprocation feed rate in mm/min Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate for retraction? Tool traversing speed 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 **Q12**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q401 Feed rate factor in %? Percentage value to which the control reduces the machining feed rate (**Q12**) as soon as the tool moves with its entire circumference within the material during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap (**Q2**) specified in Cycle **20**. The control then reduces the feed rate as per your definition at transitions and narrow places, reducing the total machining time. Input: **0.0001...100**
	Q404 Fine roughing strategy (0/1)? Define how the control will move the tool during fine roughing when the radius of the fine-roughing tool is equal to or larger than half the radius of the coarse roughing tool. **0**: Between areas that need to be fine-roughed, the control moves the tool along the contour at the current depth **1**: Between areas that need to be fine-roughed, the control retracts the tool to set-up clearance and then moves it to the starting point of the next area to be roughed out Input: **0**, **1**

Example

```
11 CYCL DEF 22 ROUGH-OUT ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q18=+0                  ;COARSE ROUGHING TOOL ~
    Q19=+0                  ;FEED RATE FOR RECIP. ~
    Q208=+99999             ;RETRACTION FEED RATE ~
    Q401=+100               ;FEED RATE FACTOR ~
    Q404=+0                 ;FINE ROUGH STRATEGY
```

15.3.28 Cycle 23 FLOOR FINISHING

ISO programming

G123

Application

With Cycle **23 FLOOR FINISHING**, you can finish your contour by taking the finishing allowance for the floor into account that has been programmed in Cycle **20**. The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically. The tool then clears the finishing allowance remaining from rough-out.

Before programming the call of Cycle **23**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable
- Cycle **22 ROUGH-OUT**, if necessary

Related topics

- Cycle **273 OCM FINISHING FLOOR** (option 167)

  Further information: "Cycle 273 OCM FINISHING FLOOR (option 167)", Page 678

Cycle sequence

1. The control positions the tool to the clearance height at rapid traverse FMAX.
2. The tool then moves in the tool axis at the feed rate **Q11**.
3. The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically
4. The tool clears the finishing allowance remaining from rough-out.
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane. There is a danger of collision!

- After the end of the cycle, position the tool with all coordinates of the working plane (e.g., **L X+80 Y+0 R0 FMAX**)
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

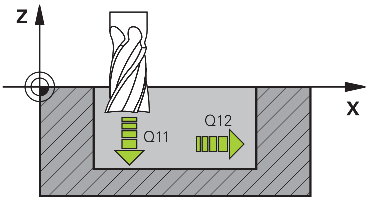
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.
- The approaching radius for pre-positioning to the final depth is permanently defined and independent of the plunging angle of the tool.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	**Q11 Feed rate for plunging?** Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate for retraction? Tool traversing speed 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 **Q12**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Example

```
11 CYCL DEF 23 FLOOR FINISHING ~
    Q11=+150              ;FEED RATE FOR PLNGNG ~
    Q12=+500              ;FEED RATE F. ROUGHNG ~
    Q208=+99999           ;RETRACTION FEED RATE
```

15.3.29 Cycle 24 SIDE FINISHING

ISO programming

G124

Application

Cycle **24 SIDE FINISHING** allows you to finish your contour by taking the side finishing allowance into account that has been programmed in Cycle **20**. You can run this cycle in climb or up-cut milling mode.

Before programming the call of Cycle **24**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable
- Cycle **22 ROUGH-OUT**, if necessary

Related topics

- Cycle **274 OCM FINISHING SIDE** (option 167)

  Further information: "Cycle 274 OCM FINISHING SIDE (option 167)", Page 681

Cycle sequence

1. The control positions the tool above the workpiece surface to the starting point for the approach position. This position in the plane results from a tangential arc on which the control moves the tool when approaching the contour
2. The control then moves the tool to the first plunging depth using the feed rate for plunging
3. The contour is approached on a tangential arc and machined up to the end. Each subcontour is finished separately
4. The tool moves on a tangential helical arc when approaching the finishing contour or retracting from it. The starting height of the helix is 1/25 of the set-up clearance **Q6**, but max. the remaining last plunging depth above the final depth
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

> The starting point calculated by the control also depends on the machining sequence. If you select the finishing cycle with the **GOTO** key and then start the NC program, the starting point can be at a different location from where it would be if you execute the NC program in the defined sequence.

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane. There is a danger of collision!

- After the end of the cycle, position the tool with all coordinates of the working plane (e.g., **L X+80 Y+0 R0 FMAX**)
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If no allowance was defined in Cycle **20**, the control generates the error message "Tool radius too large."
- If you run Cycle **24** without having roughed out with Cycle **22**, then enter "0" for the radius of the rough mill.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket and the allowance programmed in Cycle **20**.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.
- You can execute this cycle using a grinding tool.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

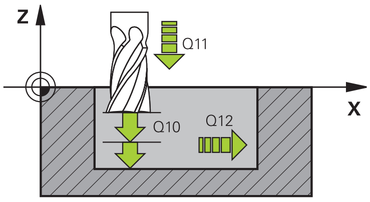
Notes on programming

- The sum of finishing allowance for the side (**Q14**) and the radius of the finish mill must be smaller than the sum of allowance for side (**Q3**, Cycle **20**) and the radius of the rough mill.
- The finishing allowance for the side **Q14** is left over after finishing. Therefore, it must be smaller than the allowance in Cycle **20**.
- Cycle **24** can also be used for contour milling. In that case, you must do the following:
  - Define the contour to be milled as a single island (without pocket boundary)
  - In Cycle **20**, enter a finishing allowance (**Q3**) greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket:
  - **PosBeforeMachining**: Return to starting position.
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	Q9 Direction of rotation? cw = -1 Machining direction: **+1**: Counterclockwise **–1**: Clockwise Input: **-1**, **+1**
 	**Q10 Plunging depth?** Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q14 Finishing allowance for side? The finishing allowance for the side **Q14** is left over after finishing. This allowance must be smaller than the allowance in Cycle **20**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **Q438 = –1**: The control assumes that the tool last used is the rough-out tool (default behavior) **Q438 = 0**: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0. Input: **-1...+32767.9** or **255** characters

Example

11 CYCL DEF 24 SIDE FINISHING ~
Q9=+1 ;ROTATIONAL DIRECTION ~
Q10=+5 ;PLUNGING DEPTH ~
Q11=+150 ;FEED RATE FOR PLNGNG ~
Q12=+500 ;FEED RATE F. ROUGHNG ~
Q14=+0 ;ALLOWANCE FOR SIDE ~
Q438=-1 ;ROUGH-OUT TOOL

15.3.30 Cycle 270 CONTOUR TRAIN DATA

ISO programming
G270

Application

You can use this cycle to specify various properties of Cycle **25 CONTOUR TRAIN**.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **270** is DEF-active, which means that it becomes effective as soon as it is defined in the NC program.
- If Cycle **270** is used, do not define any radius compensation in the contour subprogram.
- Define Cycle **270** before Cycle **25**.

Cycle parameters

Help graphic	Parameters
	Q390 Type of approach/departure? Definition of type of approach/departure: **1**: Approach the contour tangentially on a circular arc **2**: Approach the contour tangentially on a straight line **3**: Approach the contour at a right angle **0** and **4**: No approach or departure movement is performed. Input: **1**, **2**, **3**
	Q391 Radius comp. (0=R0/1=RL/2=RR)? Definition of radius compensation: **0**: Machine the defined contour without radius compensation **1**: Machine the defined contour with compensation to the left **2**: Machine the defined contour with compensation to the right Input: **0**, **1**, **2**
	Q392 App. radius/dep. radius? Only in effect if a tangential approach on a circular path was selected (**Q390** = 1). Radius of the approach/departure arc Input: **0...99999.9999**
	Q393 Center angle? Only in effect if a tangential approach on a circular path was selected (**Q390** = 1). Angular length of the approach arc Input: **0...99999.9999**
	Q394 Distance from aux. point? Only in effect if a tangential approach on a straight line or a right-angle approach is selected (**Q390** = 2 or **Q390** = 3). Distance to the auxiliary point from which the tool will approach the contour. Input: **0...99999.9999**

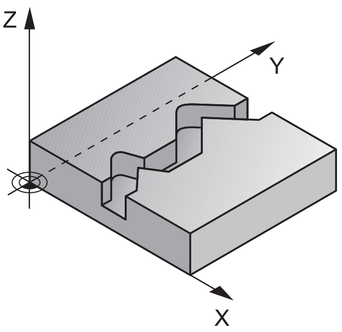
Example

```
11 CYCL DEF 270 CONTOUR TRAIN DATA ~
    Q390=+1                 ;TYPE OF APPROACH ~
    Q391=+1                 ;RADIUS COMPENSATION ~
    Q392=+5                 ;RADIUS ~
    Q393=+90                ;CENTER ANGLE ~
    Q394=+0                 ;DISTANCE
```

15.3.31 Cycle 25 CONTOUR TRAIN

ISO programming
G125

Application



In conjunction with Cycle **14 CONTOUR**, this cycle enables you to machine open and closed contours.

Cycle **25 CONTOUR TRAIN** offers considerable advantages over machining a contour using positioning blocks:

- The control monitors the operation to prevent undercuts and contour damage (run a graphic simulation of the contour before execution)
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked
- Machining can be done throughout by up-cut or by climb milling. The type of milling will even be retained if the contours were mirrored
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane. There is a danger of collision!

- After the end of the cycle, position the tool with all coordinates of the working plane (e.g., **L X+80 Y+0 R0 FMAX**)
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control takes only the first label of Cycle **14 CONTOUR** into account.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- You can execute this cycle using a grinding tool.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

Notes on programming

- Cycle **20 CONTOUR DATA**, is not required.
- 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 use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between workpiece surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q5 Workpiece surface coordinate? Absolute coordinate of the top surface of the workpiece Input: **-99999.9999...+99999.9999**
	Q7 Clearance height? Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q15 Climb or up-cut? up-cut = -1 **+1**: Climb milling **-1**: Up-cut milling **0**: Climb milling and up-cut milling alternately in several infeeds Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q18 or **QS18 Coarse roughing tool?** Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters
	Q446 Accepted residual material? Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input: **0.001...9.999**
	Q447 Maximum connection distance? Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input: **0...999.999**
	Q448 Path extension? Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour. Input: **0...99.999**

Example

```
11 CYCL DEF 25 CONTOUR TRAIN ~
    Q1=-20                  ;MILLING DEPTH ~
    Q3=+0                   ;ALLOWANCE FOR SIDE ~
    Q5=+0                   ;SURFACE COORDINATE ~
    Q7=+50                  ;CLEARANCE HEIGHT ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q15=+1                  ;CLIMB OR UP-CUT ~
    Q18=+0                  ;COARSE ROUGHING TOOL ~
    Q446=+0.01              ;RESIDUAL MATERIAL ~
    Q447=+10                ;CONNECTION DISTANCE ~
    Q448=+2                 ;PATH EXTENSION
```

15.3.32 Cycle 275 TROCHOIDAL SLOT

ISO programming
G275

Application

In conjunction with Cycle **14 KONTUR**, this cycle enables you to completely machine open and closed slots or contour slots using trochoidal milling.

With trochoidal milling, large cutting depths and high cutting speeds can be combined as the equally distributed cutting forces prevent increased wear of the tool. When indexable inserts are used, the entire cutting length is exploited to increase the attainable chip volume per tooth. Moreover, trochoidal milling is easy on the machine mechanics. Enormous amounts of time can also be saved by combining this milling method with the integrated adaptive feed control **AFC** (option 45).

Further information: "Adaptive Feed Control (AFC, option 45)", Page 1196

Depending on the cycle parameters you select, the following machining alternatives are available:

- Complete machining: Roughing, side finishing
- Only roughing
- Only side finishing

Program structure: Machining with SL Cycles

```
0 BEGIN CYC275 MM
...
12 CYCL DEF 14 CONTOUR
...
13 CYCL DEF 275 TROCHOIDAL SLOT
...
14 CYCL CALL M3
...
50 L Z+250 R0 FMAX M2
51 LBL 10
...
55 LBL 0
...
99 END PGM CYC275 MM
```

Cycle sequence

Roughing closed slots

In case of a closed slot, the contour description must always start with a straight-line block (**L** block).

1 Following the positioning logic, the tool moves to the starting point of the contour description and moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached

Finishing closed slots

5 If a finishing allowance has been defined, the control finishes the slot walls, in multiple infeeds, if so specified. Starting from the defined starting point, the control approaches the slot wall tangentially. Climb or up-cut milling is taken into consideration.

Roughing open slots

The contour description of an open slot must always start with an approach block (**APPR**).

1 Following the positioning logic, the tool moves to the starting point of the machining operation as defined by the parameters in the **APPR** block and plunges vertically to the first plunging depth.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached

Finishing open slots

5 If a finishing allowance has been defined, the control finishes the slot walls (in multiple infeeds if specified). The control approaches the slot wall starting from the defined starting point of the **APPR** block. Climb or up-cut milling is taken into consideration

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane. There is a danger of collision!

- After the end of the cycle, position the tool with all coordinates of the working plane (e.g., **L X+80 Y+0 R0 FMAX**)
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

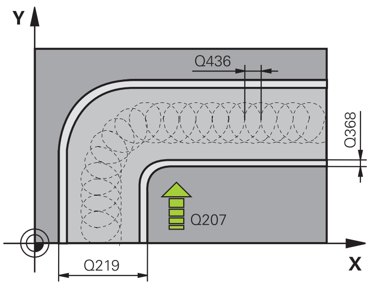
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- In conjunction with Cycle **275**, the control does not require Cycle **20 CONTOUR DATA**.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

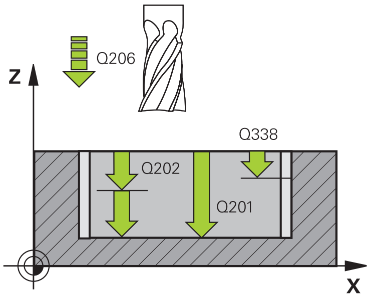
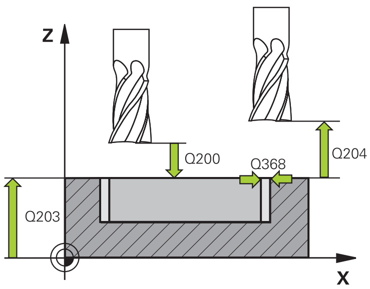
  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

Notes on programming

- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If using Cycle **275 TROCHOIDAL SLOT**, you may define only one contour subprogram in Cycle **14 CONTOUR**.
- Define the center line of the slot with all available path functions in the contour subprogram.
- The starting point of a closed slot must not be located in a contour corner.

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**
	**Q219 Width of slot?** Enter the width of the slot, which must be parallel to the secondary axis of the working plane. If the slot width equals the tool diameter, the control will mill an oblong hole. Maximum slot width for roughing: Twice the tool diameter Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q436 Feed per revolution? Value by which the control moves the tool in the machining direction per revolution. This value has an absolute 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**
	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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Help graphic	Parameter
	**Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	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**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	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**
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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** or **PREDEF**
	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** = No function **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 Input: **0**, **1**, **2** or **PREDEF**

Help graphic	Parameter
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

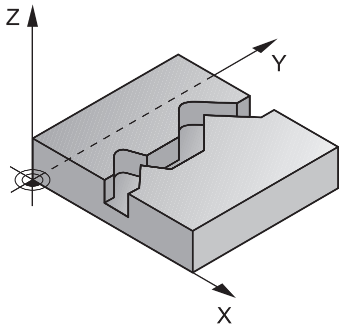
Example

```
11 CYCL DEF 275 TROCHOIDAL SLOT ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q219=+10                ;SLOT WIDTH ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q436=+2                 ;INFEED PER REV. ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q366=+2                 ;PLUNGE ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q439=+0                 ;FEED RATE REFERENCE
12 CYCL CALL
```

15.3.33 Cycle 276 THREE-D CONT. TRAIN

ISO programming
G276

Application



In conjunction with Cycle **14 CONTOUR** and Cycle **270 CONTOUR TRAIN DATA**, this cycle enables you to machine open and closed contours. You can also work with automatic residual material detection. This way you can subsequently complete for example inside corners with a smaller tool.

In contrast to Cycle **25 CONTOUR TRAIN**, Cycle **276 THREE-D CONT. TRAIN** also processes tool axis coordinates defined in the contour subprogram. This cycle can thus machine three-dimensional contours.

We recommend that you program Cycle **270 CONTOUR TRAIN DATA** before Cycle **276 THREE-D CONT. TRAIN**.

Cycle sequence

Machining a contour without infeed: Milling depth Q1 = 0

1 The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle **270 CONTOUR TRAIN DATA** (e.g., the Type of approach). The control then moves the tool to the first plunging depth
2 According to the previously defined Cycle **270 CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3 At the end of the contour, the tool will be retracted as defined in Cycle **270 CONTOUR TRAIN DATA**
4 Finally, the control retracts the tool to the clearance height.

Machining a contour with infeed: Milling depth Q1 not equal to 0 and plunging depth Q10 are defined

1 The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle **270 CONTOUR TRAIN DATA** (e.g., the Type of approach). The control then moves the tool to the first plunging depth
2 According to the previously defined Cycle **270 CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3 If you selected machining with climb milling and up-cut milling (**Q15** = 0), the control will perform a reciprocation movement. The infeed movement (plunging) will be performed at the end and at the starting point of the contour. If **Q15** is not equal to 0, the tool is moved to clearance height and is returned to the starting point of machining. From there, the control moves the tool to the next plunging depth
4 The departure will be performed as defined in Cycle **270 CONTOUR TRAIN DATA**
5 This process is repeated until the programmed depth is reached.
6 Finally, the control retracts the tool to the clearance height

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane. There is a danger of collision!

- After the end of the cycle, position the tool with all coordinates of the working plane (e.g., **L X+80 Y+0 R0 FMAX**)
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

NOTICE

Danger of collision!

A collision may occur if you position the tool behind an obstacle before the cycle is called.

- Before the cycle call, position the tool in such a way that the tool can approach the starting point of the contour without collision
- If the position of the tool is below the clearance height when the cycle is called, the control will issue an error message

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program **APPR** and **DEP** blocks for contour approach and departure, the control monitors whether the execution of any of these blocks would damage the contour.
- If using Cycle **25 CONTOUR TRAIN**, you can define only one subprogram in Cycle **14 CONTOUR**.
- We recommend that you use Cycle **270 CONTOUR TRAIN DATA** in conjunction with Cycle **276**. Cycle **20 CONTOUR DATA**, however, is not required.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

Notes on programming

- The first NC block in the contour subprogram must contain values in all of the three axes X, Y and Z.
- The algebraic sign for the depth parameter determines the working direction. If you program DEPTH = 0, the control will use the tool axis coordinates that have been specified in the contour subprogram.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between workpiece surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q7 Clearance height? Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q15 Climb or up-cut? up-cut = -1 **+1**: Climb milling **-1**: Up-cut milling **0**: Climb milling and up-cut milling alternately in several infeeds Input: **-1**, **0**, **+1**
	Q18 or **QS18 Coarse roughing tool?** Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters

Help graphic	Parameter
	Q446 Accepted residual material? Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input: **0.001...9.999**
	Q447 Maximum connection distance? Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input: **0...999.999**
	Q448 Path extension? Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour. Input: **0...99.999**

Example

```
11 CYCL DEF 276 THREE-D CONT. TRAIN ~
    Q1=-20              ;MILLING DEPTH ~
    Q3=+0               ;ALLOWANCE FOR SIDE ~
    Q7=+50              ;CLEARANCE HEIGHT ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q15=+1              ;CLIMB OR UP-CUT ~
    Q18=+0              ;COARSE ROUGHING TOOL ~
    Q446=+0.01          ;RESIDUAL MATERIAL ~
    Q447=+10            ;CONNECTION DISTANCE ~
    Q448=+2             ;PATH EXTENSION
```

15.3.34 OCM cycles

OCM cycles

General information

Refer to your machine manual.
Your machine manufacturer enables this function.

Using OCM cycles (**Optimized Contour Milling**), you can combine subcontours to form complex contours. These cycles provide more functionality than Cycles **22** to **24**. The OCM cycles feature the following additional functions:

- When roughing, the control will maintain the specified tool angle precisely
- Besides pockets, you can also machine islands and open pockets

Programming and operating notes:

- You can program up to 16 384 contour elements in one OCM cycle.
- OCM cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always verify the program graphically! This is a simple way of finding out whether the program calculated by the control will provide the desired results.

Contact angle

When roughing, the control will retain the tool angle precisely. The tool angle can be defined implicitly by specifying an overlap factor. The maximum overlap factor is 1.99; this corresponds to an angle of nearly 180°.

Contour

Specify the contour with **CONTOUR DEF / SEL CONTOUR** or with the OCM shape cycles **127x**.

Closed pockets can also be defined in Cycle **14**.

The machining dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **271 OCM CONTOUR DATA** or in the **127x** figure cycles.

CONTOUR DEF / SEL CONTOUR:

In **CONTOUR DEF / SEL CONTOUR**, the first contour can be a pocket or a boundary. The next contours can be programmed as islands or pockets. To program open pockets, use a boundary and an island.

Proceed as follows:

- Program **CONTOUR DEF**
- Define the first contour as a pocket and the second one as an island
- Define Cycle **271 OCM CONTOUR DATA**
- Program cycle parameter **Q569 = 1**
- > The control will interpret the first contour as an open boundary instead of a pocket. Thus, the open boundary and the island programmed subsequently are combined to form an open pocket.
- Define Cycle **272 OCM ROUGHING**

Programming notes:

- Subsequently defined contours that are outside the first contour will not be considered.
- The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth Therefore, start programming the subcontour with the deepest pocket.

OCM figure cycles:

The figure defined in an OCM figure cycles can be a pocket, an island, or a boundary. Use the Cycles **128x** for programming an island or an open pocket.

Proceed as follows:

- Program a figure using cycles **127x**
- If the first figure will be an island or an open pocket, make sure to program boundary cycle **128x**.
- Define Cycle **272 OCM ROUGHING**

Further information: "OCM cycles for pattern definition", Page 446

Program structure: Machining with OCM cycles

```
0 BEGIN OCM MM
...
12 CONTOUR DEF
...
13 CYCL DEF 271 OCM CONTOUR DATA
...
16 CYCL DEF 272 OCM ROUGHING
...
17 CYCL CALL
...
20 CYCL DEF 273 OCM FINISHING FLOOR
...
21 CYCL CALL
...
24 CYCL DEF 274 OCM FINISHING SIDE
...
25 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM OCM MM
```

Removing residual material

When roughing, these cycles allow you to use larger tools for the first roughing passes and then smaller tools to remove the residual material. During finishing the control will take into account the material roughed out, thus preventing the finishing tool from being overloaded.

Further information: "Example: Open pocket and fine roughing with OCM cycles", Page 729

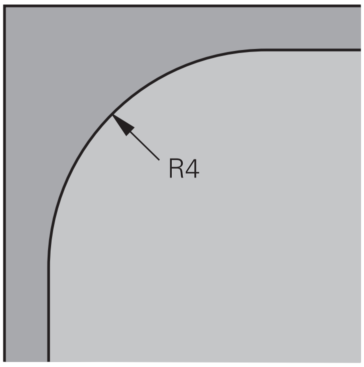
> - If residual material remains in the inside corners after roughing, then use a smaller rough-out tool or define an additional roughing operation with a smaller tool.
> - If the inside corners cannot be roughed out completely, the control may damage the contour during chamfering. In order to prevent damage to the contour, follow the procedure described below.

Procedure regarding residual material in inside corners

The example describes the inside machining of a contour by using several tools with radii greater than the programmed contour. Although the radius of the tools used becomes smaller, residual material remains in the inside corners after roughing. The control takes this residual material into account during the subsequent finishing and chamfering operations.

In the example, you use the following tools:

- **MILL_D20_ROUGH**, Ø 20 mm
- **MILL_D10_ROUGH**, Ø 10 mm
- **MILL_D6_FINISH**, Ø 6 mm
- **NC_DEBURRING_D6**, Ø 6 mm



Inside corner with a radius of 4 mm in this example

Roughing

- Rough the contour with the tool **MILL_D20_ROUGH**
- The control takes into account the Q parameter **Q578 INSIDE CORNER FACTOR**, resulting in inside radii of 12 mm during initial roughing.

...	
12 TOOL CALL Z "MILL_D20_ROUGH"	
...	
15 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR** **...**	Resulting inside radius = R_T+ (**Q578** * R_T) **10 + (0.2 *10) = 12**
16 CYCL DEF 272 OCM ROUGHING	
...	

- Then rough the contour with the smaller tool **MILL_D10_ROUGH**
- The control takes into account the Q parameter **Q578 INSIDE CORNER FACTOR**, resulting in inside radii of 6 mm during initial roughing.

...	
20 TOOL CALL Z "MILL_D10_ROUGH"	
...	
22 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR** **...**	Resulting inside radius = R_T+ (**Q578** * R_T) **5 + (0.2 *5) = 6**
23 CYCL DEF 272 OCM ROUGHING	
... **Q438 = -1 ;ROUGH-OUT TOOL** **...**	**-1**: The control assumes that the tool last used is the rough-out tool

Finishing

- Finish the contour with the tool **MILL_D6_FINISH**
- This finishing tool would allow inside radii of 3.6 mm. This means that the finishing tool would be capable of machining the defined inside radii of 4 mm. However, the control takes into account the residual material of the rough-out tool **MILL_D10_ROUGH**. The control machines the contour with the previous roughing tool's inside radii of 6 mm. Thus, the finishing cutter will be protected from overload.

...	
27 TOOL CALL Z "MILL_D6_FINISH"	
...	
29 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR** **...**	Resulting inside radius = R_T+ (**Q578** * R_T) **3 + (0.2 *3) = 3.6**
30 CYCL DEF 274 OCM FINISHING SIDE	
... **Q438 = -1 ;ROUGH-OUT TOOL** **...**	**-1**: The control assumes that the tool last used is the rough-out tool

Chamfering

- Chamfering the contour: When defining the cycle, you must define the last rough-out tool of the roughing operation.

> If you use the finishing tool as a roughing tool, the control will damage the contour. In this case, the control assumes that the finishing cutter machined the contour with inside radii of 3.6 mm. However, the finishing cutter has limited the inside radii to 6 mm based on the previous roughing operation.

...	
33 TOOL CALL Z "NC_DEBURRING_D6"	
...	
35 CYCL DEF 277 OCM CHAMFERING	
... QS438 = "MILL_D10_ROUGH" ;ROUGH-OUT TOOL ...	Rough-out tool of the last roughing operation

Positioning logic in OCM cycles

The current tool position is above the clearance height:

1. The control moves the tool to the starting point in the working plane at rapid traverse.
2. At **FMAX**, the tool moves to **Q260 CLEARANCE HEIGHT** and then to **Q200 SET-UP CLEARANCE**
3. The control then positions the tool to the starting point in the tool axis at **Q253 F PRE-POSITIONING**.

The current tool position is below the clearance height:

1. The control moves the tool to **Q260 CLEARANCE HEIGHT** at rapid traverse.
2. At **FMAX**, the tool moves to the starting point in the working plane and then to **Q200 SET-UP CLEARANCE**
3. The control then positions the tool to the starting point in the tool axis at **Q253 F PRE-POSITIONING**

> Programming and operating notes:
> - The control takes **Q260 CLEARANCE HEIGHT** from Cycle **271 OCM CONTOUR DATA** or from the figure cycles.
> - **Q260 CLEARANCE HEIGHT** is effective only if the clearance height position is above the set-up clearance.

15.3.35 Cycle 271 OCM CONTOUR DATA (option 167)

ISO programming

G271

Application

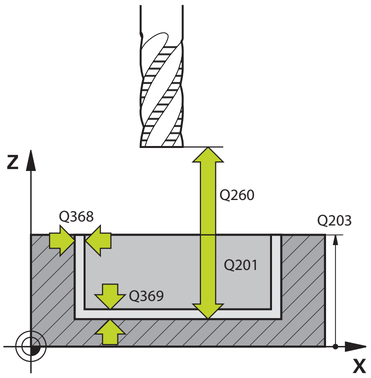
Use Cycle **271 OCM CONTOUR DATA** to program machining data for the contour or the subprograms describing the subcontours. In addition, Cycle **271** enables you to define an open boundary for a pocket.

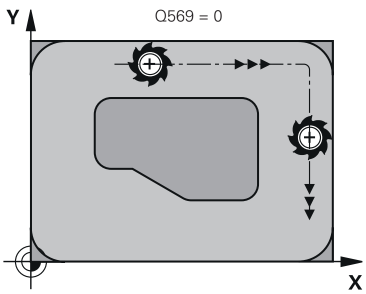
Notes

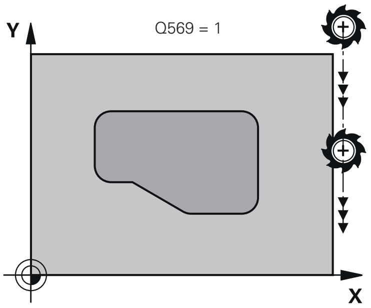
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **271** are valid for Cycles **272** to **274**.

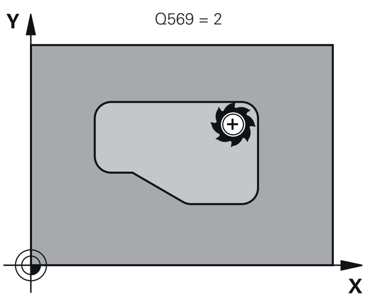
Cycle parameters

Help graphic









Parameter

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**

Q201 Depth?

Distance between the workpiece surface and the contour floor. This value has an incremental effect.

Input: **-99999.9999...+0**

Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q369 Finishing allowance for floor?

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

Q260 Clearance height?

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q578 Radius factor on inside corners?

The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**.

Input: **0.05...0.99**

Q569 Is the first pocket a boundary?

Define the boundary:

0: The first contour in **CONTOUR DEF** is interpreted as a pocket.

1: The first contour in **CONTOUR DEF** is interpreted as an open boundary. The following contour must be an island

2: The first contour in **CONTOUR DEF** is interpreted as a "bounding block." The following contour must be a pocket

Input: **0**, **1**, **2**

Example

11 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0	;SURFACE COORDINATE ~
Q201=-20	;DEPTH ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q260=+100	;CLEARANCE HEIGHT ~
Q578=+0.2	;INSIDE CORNER FACTOR ~
Q569=+0	;OPEN BOUNDARY

15.3.36 Cycle 272 OCM ROUGHING (option 167)

ISO programming
G272

Application

Use Cycle **272 OCM ROUGHING** to define the technology data for roughing.

In addition, you can use the **OCM** cutting data calculator. The calculated cutting data help to achieve high material removal rates and therefore increase the productivity.

Further information: "OCM Cutting data calculator (option 167)", Page 669

Requirements

Before programming the call of Cycle **272**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR** or Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**

Cycle sequence

1. The tool uses positioning logic to move to the starting point
2. The control determines the starting point automatically based on the pre-positioning and the programmed contour

   Further information: "Positioning logic in OCM cycles", Page 661
3. The control moves to the first plunging depth. The plunging depth and the sequence for machining the contours depend on the plunging strategy **Q575**.

   Depending on the definition in Cycle **271 OCM CONTOUR DATA**, parameter **Q569 OPEN BOUNDARY**, the control plunges as follows:
   - **Q569 = 0** or **2**: The tool plunges into the material in a helical or reciprocating movement. The finishing allowance for the side is taken into account.

     Further information: "Plunging behavior with Q569 = 0 or 2", Page 664
   - **Q569 = 1**: The tool plunges vertically outside the open boundary to the first plunging depth
4. After reaching the first plunging depth, the tool mills the contour in an outward or inward direction (depending on **Q569**) at the programmed milling feed rate **Q207**
5. In the next step, the tool is moved to the next plunging depth and repeats the roughing procedure until the programmed contour is completely machined
6. Finally, the tool retracts in the tool axis to the clearance height
7. If there are more contours, the control will repeat the machining process. The control then moves to the contour whose starting point is positioned nearest to the current tool position (depending on the infeed strategy **Q575**)
8. Finally, the tool moves to **Q200 SET-UP CLEARANCE** at **Q253 F PRE-POSITIONING** and then to **Q260 CLEARANCE HEIGHT** at **FMAX**

Plunging behavior with Q569 = 0 or 2

The control generally tries plunging with a helical path. If this is not possible, it tries plunging with a reciprocation movement.

The plunging behavior depends on:

- **Q207 FEED RATE MILLING**
- **Q568 PLUNGING FACTOR**
- **Q575 INFEED STRATEGY**
- **ANGLE**
- **RCUTS**
- $\mathbf{R_{corr}}$ (tool radius **R** + tool oversize **DR**)

Helical:

The helical path is calculated as follows:

$$Helicalradius = R_{corr} - RCUTS$$

At the end of the plunging movement, the tool executes a semi-circular movement to provide sufficient space for the resulting chips.

Reciprocating

The reciprocation movement is calculated as follows:

$$L = 2*(R_{corr} - RCUTS)$$

At the end of the plunging movement, the tool executes a linear movement to provide sufficient space for the resulting chips.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cycle does not include the corner radius **R2** in the calculation of the milling paths. Even if you use a small overlap factor, residual material may be left over on the contour floor. The residual material can cause damage to the workpiece and the tool during subsequent machining operations!

- Run a simulation to verify the machining sequence and the contour
- Use tools without a corner radius **R2** where possible

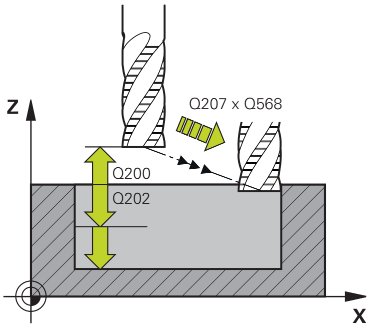
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

> If required, use a center-cut end mill (ISO 1641).

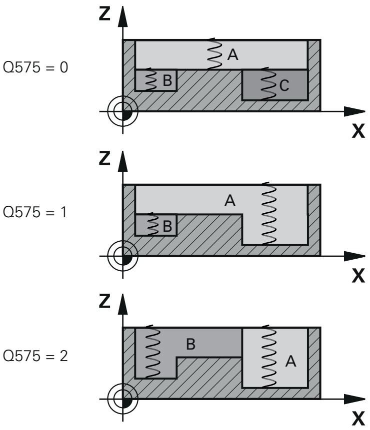
Notes on programming

- **CONTOUR DEF** / **SEL CONTOUR** will reset the tool radius that was used last. If you run this machining cycle with **Q438** = –1 after **CONTOUR DEF** / **SEL CONTOUR**, the control assumes that no pre-machining has taken place yet.
- If the path overlap factor **Q370 < 1**, a value of less than 1 is also recommended for the plunging factor **Q579** .

Cycle parameters

Help graphic	Parameter
	**Q202 Plunging depth?** Tool infeed per cut. This value has an incremental effect. Input: **0...99999.9999**
	Q370 Path overlap factor? **Q370** x tool radius = lateral infeed k on a straight line. The control maintains this value as precisely as possible. Input: **0.04...1.99** or **PREDEF**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q568 Factor for plunging feed rate? Factor by which the control reduces the feed rate **Q207** for downfeed into the material. Input: **0.1...1**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used in Cycle **272** is the rough-out tool (default behavior) **0**: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0. Input: **-1...+32767.9** or max. **255** characters

Help graphic	Parameter
	Q577 Factor for appr./dept. radius? Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input: **0.15...0.99**
	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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q576 Spindle speed? Spindle speed in revolutions per minute (rpm) for the roughing tool. **0**: The spindle speed from the **TOOL CALL** block will be used **> 0**: If a value greater than zero is entered, then this spindle speed will be used Input: **0...99999**
	Q579 Factor for plunging speed? Factor by which the control reduces the **SPINDLE SPEED Q576** for downfeed into the material. Input: **0.2...1.5**

Help graphic	Parameter
	**Q575 Infeed strategy (0/1)?** Type of downfeed: **0**: The control machines the contour from top to bottom **1**: The control machines the contour from bottom to top. The control does not always start with the deepest contour. The machining sequence is automatically calculated by the control. The total plunging path is often shorter than with strategy **2**. **2**: The control machines the contour from bottom to top. The control does not always start with the deepest contour. This strategy calculates the machining sequence such that the maximum length of the cutting edge is used. The resulting total plunging path is thus often larger than with strategy **1**. Depending on **Q568**, this may also result in a shorter machining time. Input: **0**, **1**, **2** The total plunging path is the sum of all plunging movements.

Example

```
11 CYCL DEF 272 OCM ROUGHING ~
    Q202=+5            ;PLUNGING DEPTH ~
    Q370=+0.4          ;TOOL PATH OVERLAP ~
    Q207=+500          ;FEED RATE MILLING ~
    Q568=+0.6          ;PLUNGING FACTOR ~
    Q253=+750          ;F PRE-POSITIONING ~
    Q200=+2            ;SAFETY CLEARANCE ~
    Q438=-1            ;ROUGH-OUT TOOL ~
    Q577=+0.2          ;APPROACH RADIUS FACTOR ~
    Q351=+1            ;CLIMB OR UP-CUT ~
    Q576=+0            ;SPINDLE SPEED ~
    Q579=+1            ;PLUNGING FACTOR S ~
    Q575=+0            ;INFEED STRATEGY
```

15.3.37 OCM Cutting data calculator (option 167)

Fundamentals of the OCM cutting data calculator

Introduction

The OCM cutting data calculator is used to determine the Cutting data for Cycle **272 OCM ROUGHING**. These result from the properties of the material and the tool. The calculated cutting data help to achieve high material removal rates and therefore increase the productivity.

In addition, you can use the OCM cutting data calculator to specifically influence the load on the tool via sliders for the mechanical and thermal loads. This allows you to optimize the process reliability, the wear on the tool, and the productivity.

Requirements

Refer to your machine manual!

In order to capitalize on the calculated Cutting data, you need a sufficiently powerful spindle as well as a stable machine tool.

- The entered values are based on the assumption that the workpiece is firmly clamped in place.
- The entered values are based on the assumption that the tool is seated firmly in its holder.
- The tool being used must be appropriate for the material to be machined.

In case of large cutting depths and a large angle of twist, strong pulling forces develop in the direction of the tool axis. Make sure to have a sufficient finishing allowance for the floor.

Maintaining the cutting conditions

Use the cutting data only for Cycle **272 OCM ROUGHING**.

Only this cycle ensures that the permissible tool contact angle is not exceeded for the contours to be machined.

Chip removal

NOTICE

Caution: Danger to the tool and workpiece!

If the chips are not removed in an optimum manner, they could get caught in narrow pockets at these high metal removal rates. There is then a risk of tool breakage!

- Ensure that the chips are removed in an optimum manner, as recommended by the OCM cutting data calculator.

Process cooling

The OCM cutting data calculator recommends dry cutting with cooling by compressed air for most materials. The compressed air must be aimed directly at the cutting location. The best method is through the tool holder. If this is not possible, you can also mill with an internal coolant supply.

However, chip removal might not be as efficient when using tools with an internal coolant supply. This can lead to shortened tool life.

Operation

Opening the cutting data calculator

- Select cycle **272 OCM ROUGHING**
- Select **OCM cutting data calculator** in the action bar

Closing the cutting data calculator

Apply

- Select **APPLY**
- > The control applies the determined Cutting data to the intended cycle parameters.
- > The current entries are stored, and are in place when the cutting data calculator is opened again.

or

Cancel

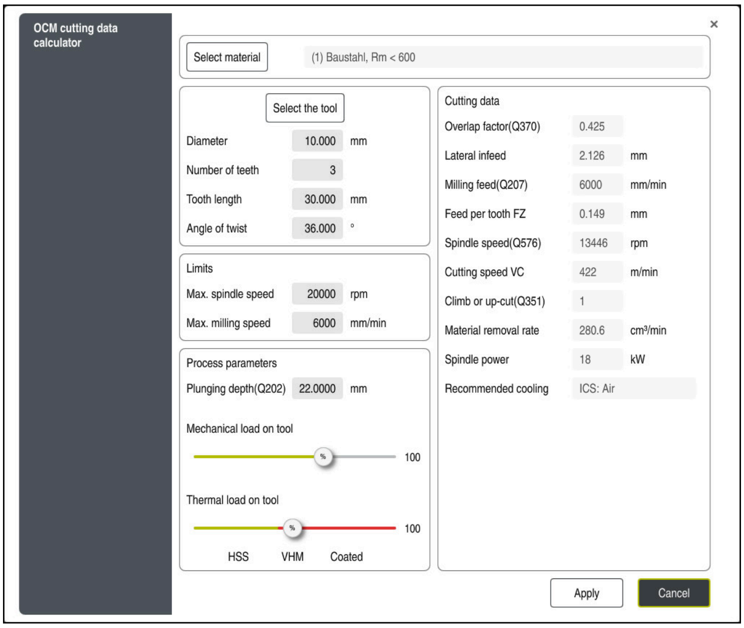
- Select **Cancel**
- > The current entries are not stored.
- > The control does not apply any values to the cycle.

The OCM cutting data calculator calculates associated values for these cycle parameters:

- Plunging depth(Q202)
- Overlap factor(Q370)
- Spindle speed(Q576)
- Climb or up-cut(Q351)

If you use the OCM cutting data calculator, then do not subsequently edit these parameters in the cycle.

Fillable form

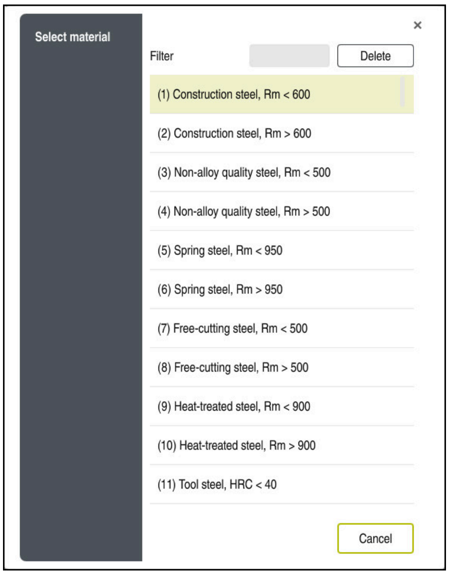


The control uses various colors and symbols in the fillable form:

- Dark gray background: entry required
- Red border of input boxes and information symbols: missing or incorrect entry
- Gray background: no entry possible

> The input field of the workpiece material is highlighted in gray. You can only select it through the selection list. The tool can also be selected through the tool table.

Workpiece material



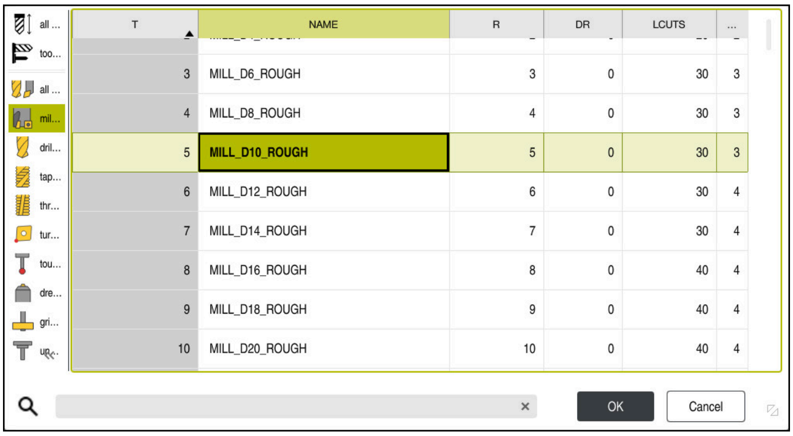
Proceed as follows to select the workpiece material:

- Select the **Select material** button
- > The control opens a selection list with various types of steel, aluminum, and titanium.
- Select the workpiece material

  or
- Enter a search term in the filter mask
- > The control displays the materials or material groups that were found. Use the **Delete** button to return to the original selection list.

> Programming and operating notes:
> - If your material is not listed in the table, choose an appropriate material group or a material with similar cutting properties
> - You will find the workpiece-material table **ocm.xml** in the **TNC:\system_calcprocess** directory

Tool



You can choose the tool either by selecting it from the tool table **tool.t** or by entering the data manually.

Proceed as follows to select the tool:

- Select the **Select the tool** button
- > The control opens the active tool table **tool.t**.
- Select the tool

  or
- Enter a tool name or number in the search field
- Confirm with **OK**
- > The control applies the **Diameter**, the **Number of teeth** and the **Tooth length** from the **tool.t** table.
- Define the **Angle of twist**

Proceed as follows to select the tool:

- Enter the **Diameter**
- Define the **Number of teeth**
- Enter the **Tooth length**
- Define the **Angle of twist**

Input dialog	Description
Diameter	Diameter of the roughing tool in mm Value is applied automatically after the roughing tool has been selected. Input: **1...40**
Number of teeth	Number of teeth of the roughing tool Value is applied automatically after the roughing tool has been selected. Input: **1...10**
Angle of twist	Angle of twist of the roughing tool in ° If there are different angles of twist, then enter the average value. Input: **0...80**

> Programming and operating notes:
> - You can modify the values of the **Diameter**, the **Number of teeth** and the **Tooth length** at any time. The modified value is **not** written to the tool table **tool.t**!
> - You will find the Angle of twist in the description of your tool, for example in the tool catalog of the tool manufacturer.

Limits

For the Limits, you need to define the maximum spindle speed and the maximum feed rate for milling. The calculated Cutting data are then limited to these values.

Input dialog	Description
Max. spindle speed	Maximum spindle speed in rpm permitted by the machine and the clamping situation: Input: **1...99999**
Max. milling speed	Maximum milling speed (feed rate) in mm/min permitted by the machine and the clamping situation: Input: **1...99999**

Process parameters

For the Process parameters, you need to define the Plunging depth(Q202) as well as the mechanical and thermal loads:

Input dialog	Description
Plunging depth(Q202)	Plunging depth (>0 mm to [6 times the tool diameter]) The value from cycle parameter **Q202** is applied when starting the OCM cutting data calculator. Input: **0.001...99999.999**,
Mechanical load on tool	Slider for selection of the mechanical load (the value is normally between 70 % and 100 %) Input: **0%...150%**
Thermal load on tool	Slider for selection of the thermal load Set the slider according to the thermal wear-resistance (coating) of your tool. ■ HSS: low thermal wear-resistance ■ VHM (uncoated or normally-coated solid carbide milling cutters): medium thermal wear-resistance ■ Coated (fully-coated solid carbide milling cutters): high thermal wear-resistance ■ The slider is only effective in the range with a green background. This limiting depends on the maximum spindle speed, the maximum feed rate, and the selected material. ■ If the slider is in the red range, the control will use the maximum permissible value. Input: **0%...200%**

Further information: "Process parameters ", Page 676

Cutting data

The control displays the calculated values in the Cutting data section.

The following Cutting data are applied to the appropriate cycle parameters in addition to the plunging depth **Q202**:

Cutting data:	Applied to cycle parameter:
Overlap factor(Q370)	**Q370** = **TOOL PATH OVERLAP**
Milling feed(Q207) in mm/min	**Q207** = **FEED RATE MILLING**
Spindle speed(Q576) in rpm	**Q576** = **SPINDLE SPEED**
Climb or up-cut(Q351)	**Q351**= **CLIMB OR UP-CUT**

Programming and operating notes:

- The OCM cutting data calculator calculates values only for climb milling **Q351**=+1. For this reason, it always applies **Q351**=+1 to the cycle parameter.
- The OCM cutting data calculator compares the cutting data with the input ranges of the cycle. If the values fall below or exceed the input ranges, the parameter will be highlighted in red in the OCM cutting data calculator. In this case, the cutting data cannot be transferred to the cycle.

The following cutting data is for informational purposes and recommendation:

- Lateral infeed in mm
- Tooth feed FZ in mm
- Cutting speed VC in m/min
- Material removal rate in cm^3/min
- Spindle power in kW
- Recommended cooling

These values help you assess whether your machine tool is able to meet the selected cutting conditions.

Process parameters

The two sliders for the mechanical and thermal load have an influence on the process forces and temperatures prevalent on the cutting edges. Higher values increase the metal removal rate, but also lead to a higher load. Moving the sliders makes different process parameters possible.

Maximum material removal rate

For a maximum material removal rate, set the slider for the mechanical load to 100 % and the slider for the thermal load according to the coating of your tool.

If the defined limitations permit it, the cutting data utilize the tool at its mechanical and thermal load capacities. For large tool diameters (D>=16 mm), a very high level of spindle power can be necessary.

For the theoretically expectable spindle power, refer to the cutting data output.

> If the permissible spindle power is exceeded, you can first move the slider for the mechanical load to a lower value. If necessary, you can also reduce the plunging depth (a_p).
>
> Please note that at very high shaft speeds, a spindle running below its rated speed will not attain the rated power.
>
> If you wish to achieve a high material removal rate, you must ensure that chips are removed optimally.

Reduced load and low wear

In order to decrease the mechanical load and the thermal wear, reduce the mechanical load to 70 %. Reduce the thermal load to a value that corresponds to 70 % of the coating of your tool.

These settings utilize the tool in a manner that is mechanically and thermally balanced. In general the tool will then reach its maximum service life. The lower mechanical load makes a smoother process possible that is less subject to vibration.

Achieving an optimum result

If the Cutting data do not lead to a satisfactory cutting process, then different causes might be the reason for this.

Excessively high mechanical load

If there is an excessive mechanical load, you must first reduce the process force.

The following conditions are indications of excessive mechanical load:

- Cutting edges of the tool break
- Shaft of the tool breaks
- Excessive spindle torque or spindle power
- Excessive axial or radial forces on the spindle bearing
- Undesired oscillations or chatter
- Oscillations due to weak clamping
- Oscillations due to long projecting tool

Excessively high thermal load

If there is an excessive thermal load, you must reduce the process temperature.

The following conditions indicate an excessive thermal load on the tool:

- Excessive crater wear at the cutting surface
- The tool glows
- The cutting edges melt (for materials that are very difficult to cut, such as titanium)

Material removal rate is too low

If the machining time is too long and it must be reduced, the material removal rate can be increased by moving both sliders.

If both the machine and the tool still have potential, then it is recommended that the slider for the process temperature be raised to a higher value first. Subsequently, if possible, you can also raise the slider for the process forces to a higher value.

Remedies for problems

The table below provides an overview of possible types of problems as well as countermeasures for them.

Condition	Slider Mechanical load on tool	Slider Thermal load on tool	Miscellaneous
Vibrations (such as weak clamping or tools that project too far)	Decrease	Perhaps increase	Check the clamping
Undesired vibrations or chatter	Decrease	-	
Shaft of tool breaks	Decrease	-	Check the chip removal
Cutting edges of the tool break	Decrease	-	Check the chip removal
Excessive wear	Perhaps increase	Decrease	
The tool glows	Perhaps increase	Decrease	Check the cooling
Machining time is too long	Perhaps increase	Increase this first	
Excessive spindle load	Decrease	-	
Excessive axial force on spindle bearing	Decrease	-	■ Reduce the plunging depth ■ Use a tool with a lower angle of twist
Excessive radial force on spindle bearing	Decrease	-	

15.3.38 Cycle 273 OCM FINISHING FLOOR (option 167)

ISO programming
G273

Application

With Cycle **273 OCM FINISHING FLOOR**, you can program finishing with the finishing allowance for the floor programmed in Cycle **271**.

Requirements

Before programming the call of Cycle **273**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable

Cycle sequence

1. The tool uses positioning logic to move to the starting point
   Further information: "Positioning logic in OCM cycles", Page 661
2. The tool then moves in the tool axis at the feed rate **Q385**
3. The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically
4. The tool mills off the material remaining from rough-out (finishing allowance)
5. Finally, the tool moves to **Q200 SET-UP CLEARANCE** at **Q253 F PRE-POSITIONING** and then to **Q260 CLEARANCE HEIGHT** at **FMAX**

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cycle does not include the corner radius **R2** in the calculation of the milling paths. Even if you use a small overlap factor, residual material may be left over on the contour floor. The residual material can cause damage to the workpiece and the tool during subsequent machining operations!

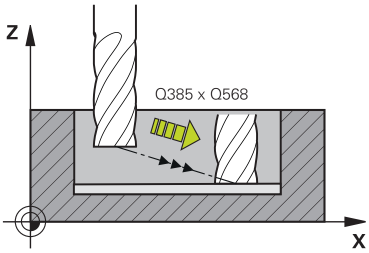
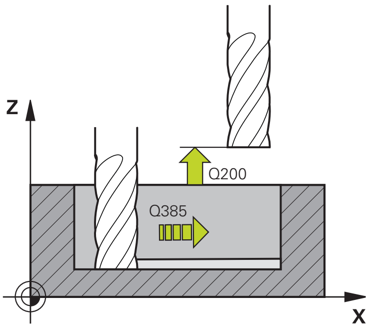
- ▶ Run a simulation to verify the machining sequence and the contour
- ▶ Use tools without a corner radius **R2** where possible

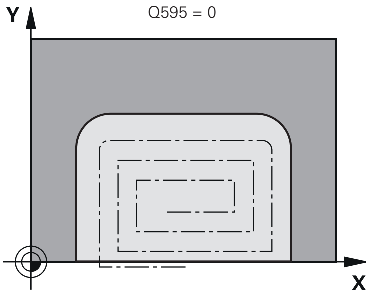
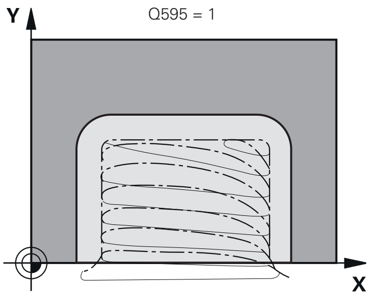
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour.
- For finishing with Cycle **273**, the tool always works in climb milling mode.
- 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.

Note on programming

- If you use an overlap factor greater than 1, residual material may be left over. Check the contour using the program verification graphics and slightly change the overlap factor, if necessary. This allows another distribution of cuts, which often provides the desired results.

Cycle parameters

Help graphic	Parameter
	Q370 Path overlap factor? **Q370** x tool radius = lateral infeed k. The overlap is considered to be the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input: **0.0001...1.9999** or **PREDEF**
	**Q385 Finishing feed rate?** Traversing speed of the tool in mm/min for floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q568 Factor for plunging feed rate? Factor by which the control reduces the feed rate **Q385** for downfeed into the material. Input: **0.1...1**
	**Q253 Feed rate for pre-positioning?** Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters

Help graphic	Parameter
 	**Q595 Strategy (0/1)?** Machining strategy for finishing **0**: Equidistant strategy = constant distances between paths **1**: Strategy with constant contact angle Input: **0**, **1**
	Q577 Factor for appr./dept. radius? Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input: **0.15...0.99**

Example

11 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+1	;TOOL PATH OVERLAP ~
Q385=+500	;FINISHING FEED RATE ~
Q568=+0.3	;PLUNGING FACTOR ~
Q253=+750	;F PRE-POSITIONING ~
Q200=+2	;SET-UP CLEARANCE ~
Q438=-1	;ROUGH-OUT TOOL ~
Q595=+1	;STRATEGY ~
Q577=+0.2	;APPROACH RADIUS FACTOR

15.3.39 Cycle 274 OCM FINISHING SIDE (option 167)

ISO programming
G274

Application

With Cycle **274 OCM FINISHING SIDE**, you can program finishing with the side finishing allowance programmed in Cycle **271**. You can run this cycle in climb or up-cut milling.

Cycle **274** can also be used for contour milling.

Proceed as follows:

- Define the contour to be milled as a single island (without pocket boundary)
- Enter the finishing allowance (**Q368**) in Cycle **271** to be greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Requirements

Before programming the call of Cycle **274**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable

Cycle sequence

1. The tool uses positioning logic to move to the starting point
2. The control positions the tool above the workpiece surface to the starting point for the approach position. This position in the plane results from a tangential arc on which the control moves the tool when approaching the contour

   Further information: "Positioning logic in OCM cycles", Page 661
3. The control then moves the tool to the first plunging depth using the feed rate for plunging
4. The tool approaches and moves along the contour helically on a tangential arc until the entire contour is finished. Each subcontour is finished separately
5. Finally, the tool moves to **Q200 SET-UP CLEARANCE** at **Q253 F PRE-POSITIONING** and then to **Q260 CLEARANCE HEIGHT** at **FMAX**

Notes

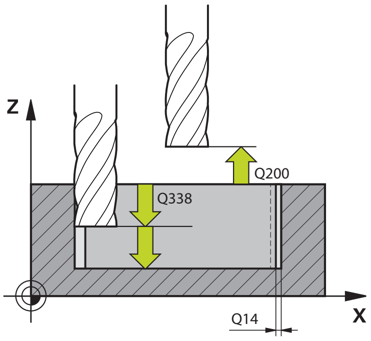
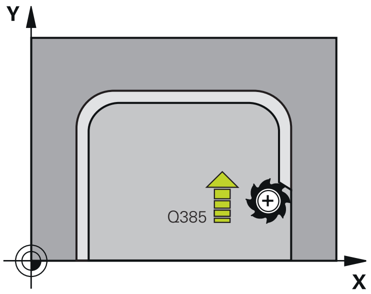
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour and the allowance programmed in Cycle **271**.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- You can execute this cycle using a grinding tool.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

Note on programming

- The finishing allowance for the side **Q14** is left over after finishing. It must be smaller than the allowance in Cycle **271**.

Cycle parameters

Help graphic	Parameter
	**Q338 Infeed for finishing?** Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q14 Finishing allowance for side? The finishing allowance for the side **Q14** is left over after finishing. This allowance must be smaller than the allowance in Cycle **271**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters
	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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Example

11 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0	;INFEED FOR FINISHING ~
Q385=+500	;FINISHING FEED RATE ~
Q253=+750	;F PRE-POSITIONING ~
Q200=+2	;SET-UP CLEARANCE ~
Q14=+0	;ALLOWANCE FOR SIDE ~
Q438=-1	;ROUGH-OUT TOOL ~
Q351=+1	;CLIMB OR UP-CUT

15.3.40 Cycle 277 OCM CHAMFERING (option 167)

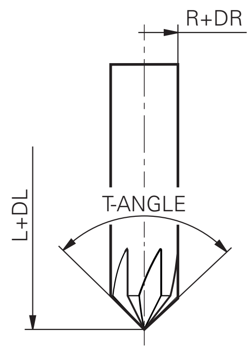
ISO programming
G277

Application

Cycle **277 OCM CHAMFERING** enables you to deburr edges of complex contours that you roughed out using OCM cycles.

This cycle considers adjacent contours and boundaries that you called before with Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements.

Requirements



Before the control can execute Cycle **277**, you need to create the tool in the tool table using appropriate parameters:

- **L** + **DL**: Overall length up to the theoretical tip
- **R** + **DR**: Definition of the overall tool radius
- **T-ANGLE**: Point angle of the tool

In addition, you need to program other cycles before programming the call of Cycle **277**:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable

Cycle sequence

1. The tool uses positioning logic to move to the starting point. This point is determined automatically based on the programmed contour

   Further information: "Positioning logic in OCM cycles", Page 661
2. In the next step, the tool moves at **FMAX** to set-up clearance **Q200**
3. Then, the tool plunges vertically to **Q353 DEPTH OF TOOL TIP**
4. The tool approaches the contour in a tangential or vertical movement (depending on the available space). For machining the chamfer, the tool uses the milling feed rate **Q207**
5. Then, the tool is retracted from the contour in a tangential or vertical movement (depending on the available space).
6. If there are several contours, the control positions the tool at clearance height after each contour and then moves it to the next starting point. Steps 3 to 6 are repeated until the programmed contour is completely chamfered
7. Finally, the tool moves to **Q200 SET-UP CLEARANCE** at **Q253 F PRE-POSITIONING** and then to **Q260 CLEARANCE HEIGHT** at **FMAX**

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for chamfering. The starting point depends on the available space.
- The control monitors the tool radius. Adjacent walls machined with Cycle **271 OCM CONTOUR DATA** or with the **12xx** figure cycles will remain intact.
- The cycle monitors for damage to the contour floor from the tool tip. This tool tip results from the radius **R**, the radius of the tool tip **R_TIP**, and the point angle **T-ANGLE**.
- Keep in mind that the active tool radius of the chamfering tool must be smaller than or equal to the radius of the rough-out tool. Otherwise, the control might not be able to completely chamfer all edges. The effective tool radius is the radius of the cutting length of the tool. This tool radius results from **T-ANGLE** and **R_TIP** from the tool table.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

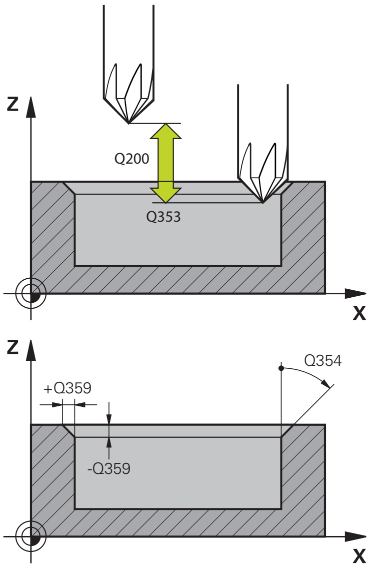
  Further information: "Adapting the feed rate for circular paths with M109", Page 1331
- If the roughing operations have not completely removed the material before chamfering, you need to define the last roughing tool in **QS438 ROUGH-OUT TOOL**, in order to prevent damage to the contour.

  "Procedure regarding residual material in inside corners"

Note on programming

- If the value of parameter **Q353 DEPTH OF TOOL TIP** is less than the value of parameter **Q359 CHAMFER WIDTH**, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	**Q353 Depth of tool tip?** Distance between theoretical tool tip and workpiece surface coordinate. This value has an incremental effect. Input: **-999.9999...-0.0001**
	Q359 Width of chamfer (-/+)? Width or depth of chamfer: **-**: Depth of chamfer **+**: Width of chamfer This value has an incremental effect. Input: **-999.9999...+999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for positioning Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters
	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 **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Help graphic	Parameter
	Q354 Angle of chamfer? Angle of the chamfer **0**: The chamfer angle is half the defined **T-ANGLE** from the tool table **> 0**: The chamfer angle is compared to the value of **T-ANGLE** from the tool table. If these two values do not match, the control will display an error message. Input: **0...89**

Example

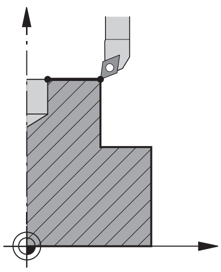
```
11 CYCL DEF 277 OCM CHAMFERING ~
    Q353=-1                 ;DEPTH OF TOOL TIP ~
    Q359=+0.2               ;CHAMFER WIDTH ~
    Q207=+500               ;FEED RATE MILLING ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q438=-1                 ;ROUGH-OUT TOOL ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q354=+0                 ;CHAMFER ANGLE
```

15.3.41 Cycle 291 COUPLG.TURNG.INTERP. (option 96)

ISO programming
G291

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.



Cycle **291 COUPLG.TURNG.INTERP.** couples the tool spindle to the position of the linear axes, or cancels this spindle coupling. With interpolation turning, the cutting edge is oriented to the center of a circle. The center of rotation is defined in the cycle by entering the coordinates **Q216** and **Q217**.

Cycle sequence

Q560=1:

1 The control first performs a spindle stop (**M5**).
2 The control orients the tool spindle to the specified center of rotation. The specified angle for spindle orientation **Q336** is taken into account. If an "ORI" value is given in the tool table, it is also taken into account.
3 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4 To terminate the cycle, the coupling must be deactivated by the operator. (With Cycle **291** or end of program/internal stop.)

Q560=0:

1 The control deactivates the spindle coupling.
2 The tool spindle is no longer coupled to the position of the linear axes.
3 The control ends machining with Cycle **291** COUPLG.TURNG.INTERP.
4 If **Q560**=0, parameters **Q336**, **Q216**, **Q217** are not relevant

Notes

This cycle is effective only for machines with servo-controlled spindle.

Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine manufacturer for further information.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **291** is CALL-active.
- This cycle can also be used in a tilted working plane.
- Remember that the axis angle must be equal to the tilt angle before the cycle call! Only then can the axis be correctly coupled.
- If Cycle **8 MIRRORING** is active, the control does **not** execute the interpolation turning cycle.
- If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.

Notes on programming

- Programming of M3/M4 is not required. To describe the circular motions of the linear axes, you can, for example, use **CC** and **C** blocks.
- When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.
- Program outside contours with a radius greater than 0.
- Program inside contours with a radius greater than the tool radius.
- In order to attain high contouring speeds for your machine, define a large tolerance with Cycle **32** before calling the cycle. Program Cycle **32** with HSC filter=1.
- After defining Cycle **291** and **CYCL CALL**, program the operation you wish to perform. To describe the circular motions of the linear axes, you can use linear or polar coordinates, for example.

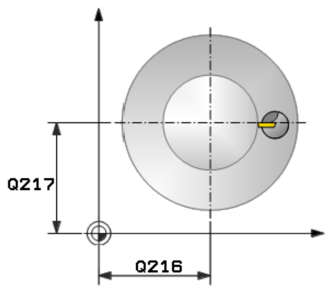
  Further information: "Example: Interpolation turning with Cycle 291", Page 739

Note regarding machine parameters

- In the machine parameter **mStrobeOrient** (no. 201005), the machine manufacturer defines the M function for spindle orientation.
  - If the value is > 0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine manufacturer). The control waits until the oriented spindle stop has been completed.
  - If you enter –1, the control will perform the oriented spindle stop.
  - If you enter 0, no action will be taken.

  The control will, under no circumstances, output **M5** before.

Cycle parameters

Help graphic	Parameter
	Q560 Spindle coupling (0=off, 1=on)? Define whether the tool spindle will be coupled to the position of the linear axes. If spindle coupling is active, the tool's cutting edge is oriented to the center of rotation. **0**: Spindle coupling off **1**: Spindle coupling on Input: **0**, **1**
	Q336 Angle for spindle orientation? The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that one cutting edge is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input: **0...360** **Further information:** "Defining the tool", Page 690
	**Q216 Center in 1st axis?** Center of rotation in the main axis of the working plane Absolute input: **-99999.9999...99999.9999**
	Q217 Center in 2nd axis? Center of rotation in the secondary axis of the working plane Input: **-99999.9999...+99999.9999**
	Q561 Convert turning tool (0/1) Only relevant if you define the turning tool in the turning tool table (toolturn.trn). This parameter allows you to decide whether the value XL of the turning tool will be interpreted as radius R of a milling tool. **0**: No change; the turning tool is interpreted as described in the turning tool table (toolturn.trn). In this case, you must not use the radius compensation **RR** or **RL**. Furthermore, you must describe the movement of the path of the tool center point **TCP** without spindle coupling when programming. This kind of programming is much more complicated. **1:** The value XL from the turning tool table (toolturn.trn) is interpreted as a radius R of a milling tool table. This makes it possible to use radius compensation **RR** or **RL** when programming your contour. This kind of programming is recommended. Input: **0**, **1**

Example

```
11 CYCL DEF 291 COUPLG.TURNG.INTERP. ~
    Q560=+0                    ;SPINDLE COUPLING ~
    Q336=+0                    ;ANGLE OF SPINDLE ~
    Q216=+50                   ;CENTER IN 1ST AXIS ~
    Q217=+50                   ;CENTER IN 2ND AXIS ~
    Q561=+0                    ;CONVERT FROM TURNING TOOL
```

Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either activate (**Q560**=1) or deactivate (**Q560**=0) the COUPLG.TURNG.INTERP. cycle.

Spindle coupling off, Q560=0

The tool spindle is not coupled to the position of the linear axes.

> **Q560**=0: Disable the **COUPLG.TURNG.INTERP.** cycle!

Spindle coupling on, Q560=1

A turning operation is executed with the tool spindle coupled to the position of the linear axes. If you set the parameter **Q560**=1, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). The geometry data of the turning tool are converted to the data of a milling cutter. Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

  NOTICE

  Danger of collision!

  Collision may occur between the tool holder and workpiece during inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

  ▶ Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

- **Define a turning tool in the turning tool table (toolturn.trn)**

  If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case, the orientation of the spindle to the center of rotation takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table), parameter **Q336**, and parameter **Q561**.

Programming and operating notes:

- If you define the turning tool in the turning tool table (toolturn.trn), we recommend working with parameter **Q561**=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. With **Q561**=1 you can use radius compensation **RR** and **RL** when programming. (However, if you program **Q561**=0, then you cannot use radius compensation **RR** and **RL** when describing your contour. Additionally, you must program the movement of the tool center path **TCP** without spindle coupling. This kind of programming is much more complicated!)

  If you programmed parameter **Q561**=1, you must program the following in order to conclude the interpolation turning machining operation:
  - R0, cancels radius compensation
  - Cycle **291** with parameters **Q560**=0 and **Q561**=0, deactivates spindle coupling
  - **CYCL CALL**, for calling Cycle **291**
  - **TOOL CALL** overrides the conversion of parameter **Q561**

  If you programmed parameter **Q561**=1, you may only use the following types of tools:
  - **TYPE**: **ROUGH**, **FINISH**, **BUTTON** with the machining directions **TO**: 1 or 8, **XL**>=0
  - **TYPE**: **ROUGH**, **FINISH**, **BUTTON** with the machining directions **TO**: 7: **XL**<=0

The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	**ORI** + **Q336**
Interpolation turning, inside	7	**ORI** + **Q336** + 180
Interpolation turning, outside	7	**ORI** + **Q336** + 180
Interpolation turning, inside	1	**ORI** + **Q336**
Interpolation turning, outside	8	**ORI** + **Q336**
Interpolation turning, inside	8	**ORI** + **Q336**

You can use the following tool types for interpolation turning:

- TYPE: ROUGH, with the machining directions TO: 1, 7, 8
- TYPE: FINISH, with the machining directions TO: 1, 7, 8
- TYPE: BUTTON, with the machining directions TO: 1, 7, 8

The following tool types cannot be used for interpolation turning:

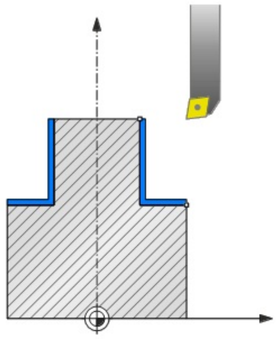
- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD

15.3.42 Cycle 292 CONTOUR.TURNG.INTRP. (option 96)

ISO programming
G292

Application

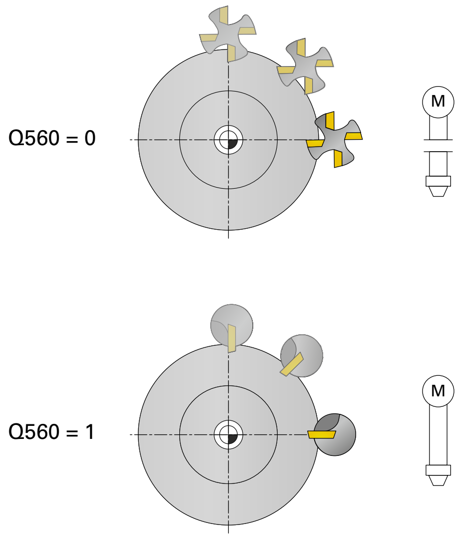
⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



Cycle **292 INTERPOLATION TURNING CONTOUR FINISHING** couples the tool spindle to the positions of the linear axes. This cycle enables you to machine specific rotationally symmetrical contours in the active working plane. You can also run this cycle in the tilted working plane. The center of rotation is the starting point in the working plane at the time the cycle is called. After executing this cycle, the control deactivates the spindle coupling again.

Before using Cycle **292**, you first need to define the desired contour in a subprogram and reference this contour with Cycle **14** or **SEL CONTOUR**. Program the contour either with monotonically decreasing or monotonically increasing coordinates. Undercuts cannot be machined with this cycle. If you enter **Q560**=1, you can turn the contour and the cutting edge is oriented toward the circle center. If you enter **Q560**=0, you can mill the contour and the spindle is not oriented toward the circle center.

Cycle sequence



Cycle Q560=0: Contour milling

1 The M3/M4 function programmed before the cycle call remains in effect.
2 No spindle stop and **no** spindle orientation will be performed. **Q336** is not taken into account
3 The control positions the tool at the contour start radius **Q491**, taking the selected machining type (inside/outside, Q529) and the set-up clearance to the side (**Q357**) into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4 The control machines the defined contour using a rotating spindle (M3/M4). The principal axes of the working plane move along a circular path, whereas the spindle axis does not follow.
5 At the end point of the contour, the control retracts the tool perpendicularly to set-up clearance.
6 Finally, the control retracts the tool to the clearance height.

Cycle Q560=1: Contour turning

1 The control orients the tool spindle to the specified center of rotation. The specified angle **Q336** is taken into account. If an "ORI" value is given in the turning tool table (toolturn.trn), it is also taken into account.
2 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
3 The control positions the tool at the contour start radius **Q491**, taking the selected machining type (inside/outside, **Q529**) and the set-up clearance to the side (**Q357**) into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4 The control uses the interpolation turning cycle to machine the defined contour. In interpolation turning, the linear axes of the working plane move along a circular path, whereas the spindle axis follows, it is oriented perpendicularly to the surface.
5 At the end point of the contour, the control retracts the tool perpendicularly to set-up clearance.
6 Finally, the control retracts the tool to the clearance height.
7 The control automatically deactivates the coupling of the tool spindle to the linear axes.

Notes

This cycle is effective only for machines with servo-controlled spindle.
Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine manufacturer for further information.

NOTICE

Danger of collision!

There is a risk of collision between tool and workpiece. The control does not automatically extend the described contour by a set-up clearance! At the beginning of the machining operation, the control positions the tool at rapid traverse FMAX to the contour starting point!

- Program an extension of the contour in the subprogram
- Make sure that there is no material at the contour starting point
- The center of the turning contour is the starting point in the working plane at the time the cycle is called

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The cycle is CALL-active.
- Roughing operations with multiple passes are not possible in this cycle.
- For inside contours, the control checks whether the active tool radius is less than half the diameter at the start of contour **Q491** plus the set-up clearance to the side **Q357**. If the control determines that the tool is too large, the NC program will be canceled.
- Remember that the axis angle must be equal to the tilt angle before the cycle call! Only then can the axis be correctly coupled.
- If Cycle **8 MIRRORING** is active, the control does **not** execute the interpolation turning cycle.
- If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.
- In parameter **Q449 FEED RATE**, you program the feed rate at the starting radius. Keep in mind that the feed rate in the status display is referenced to the **TCP** and may deviate from **Q449**. The control calculates the feed rate in the status display as follows.

  Outside machining **Q529 = 1**

  $$F_{TCP} = Q449 \times \frac{(Q491 + R)}{Q491}$$

  Inside machining **Q529 = 0**

  $$F_{TCP} = Q449 \times \frac{(Q491 - R)}{Q491}$$

Notes on programming

- Program the turning contour without tool radius compensation (RR/RL) and without APPR or DEP movements.
- Please note that it is not possible to define programmed finishing allowances via the **FUNCTION TURNDATA CORR-TCS(WPL)** function. Program a finishing allowance for your contour directly in the cycle or by specifying a tool compensation (DXL, DZL, DRS) in the tool table.
- When programming, remember to use only positive radius values.
- When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.
- Program outside contours with a radius greater than 0.
- Program inside contours with a radius greater than the tool radius.
- In order to attain high contouring speeds for your machine, define a large tolerance with Cycle **32** before calling the cycle. Program Cycle **32** with HSC filter=1.
- If you deactivate the spindle coupling (**Q560 = 0**), you can execute this cycle with polar kinematics. This requires that you clamp the workpiece at the center of the rotary table.

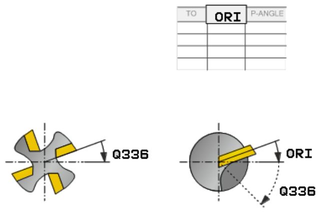
  Further information: "Machining with polar kinematics with FUNCTION POLARKIN", Page 1295

Note regarding machine parameters

- With **Q560**=1, the control does not check whether the cycle is run with a rotating or stationary spindle. (Independent of **CfgGeoCycle** - **displaySpindleError** (no. 201002))
- In the machine parameter **mStrobeOrient** (no. 201005), the machine manufacturer defines the M function for spindle orientation.
  - If the value is > 0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine manufacturer). The control waits until the oriented spindle stop has been completed.
  - If you enter –1, the control will perform the oriented spindle stop.
  - If you enter 0, no action will be taken.

  The control will, under no circumstances, output **M5** before.

Cycle parameters

Help graphic	Parameter
	Q560 Spindle coupling (0=off, 1=on)? Define whether the spindle will be coupled or not. **0**: Spindle coupling off (mill the contour) **1**: Spindle coupling on (turn the contour) Input: **0...1**
	**Q336 Angle for spindle orientation?** The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that one cutting edge is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input: **0...360**
	Q546 Reverse tool rotation direction? Direction of spindle rotation of the active tool: **3**: Clockwise rotating tool (M3) **4**: Counter-clockwise rotating tool (M4) Input: **3**, **4**
	Q529 Machining operation (0/1)? Define whether an inside or outside contour will be machined: **+1**: Inside machining **0**: Outside machining Input: **0**, **1**
	Q221 Oversize for surface? Allowance in the working plane Input: **0...99.999**
	Q441 Infeed per revolution [mm/rev]? Dimension by which the control moves the tool during one revolution. Input: **0,001...99.999**
	Q449 Feed rate / cutting speed? (mm/min) Feed rate relative to the contour starting point **Q491**. The feed rate of the tool center point path is adjusted depending on the tool radius and **Q529 MACHINING OPERATION**. From these parameters, the control determines the programmed cutting speed at the diameter of the contour starting point. **Q529 = 1**: Feed rate of the tool center point path is reduced for inside machining. **Q529 = 0**: Feed rate of the tool center point path is increased for outside machining. Input: **1...99999** or **FAUTO**

Help graphic	Parameter
	Q491 Contour starting point (radius)? Radius of the contour starting point (e.g., X coordinate, if tool axis is Z). This value has an absolute effect. Input: **0.9999...99999.9999**
	Q357 Safety clearance to the side? Set-up clearance to the side of the workpiece when the tool approaches the first plunging depth. This value has an incremental effect. Input: **0...99999.9999**
	Q445 Clearance height? Absolute height at which collision between tool and workpiece is impossible. The tool retracts to this position at the end of the cycle. Input: **-99999.9999...+99999.9999**
	Q592 Type of dimension (0/1)? Interpretation of the contour dimensions: **0**: The control interprets the contour in the **ZX** coordinate plane. The control interprets the X axis values as radii. The coordinate system is left-handed. Therefore, the programmed direction of rotation for circles is as follows: ■ **DR-**: In clockwise direction ■ **DR+**: In counterclockwise direction **1**: The control interprets the contour in the **ZXØ** coordinate plane. The control interprets the X axis values as diameters. The coordinate system is right-handed. Therefore, the programmed direction of rotation for circles is as follows: ■ **DR-**: In counterclockwise direction ■ **DR+**: In clockwise direction Input: **0**, **1**

Example

```
11 CYCL DEF 292 CONTOUR.TURNG.INTRP. ~
    Q560=+0                 ;SPINDLE COUPLING ~
    Q336=+0                 ;ANGLE OF SPINDLE ~
    Q546=+3                 ;CHANGE TOOL DIRECTN. ~
    Q529=+0                 ;MACHINING OPERATION ~
    Q221=+0                 ;SURFACE OVERSIZE ~
    Q441=+0.3               ;INFEED ~
    Q449=+2000              ;FEED RATE ~
    Q491=+50                ;CONTOUR START RADIUS ~
    Q357=+2                 ;CLEARANCE TO SIDE ~
    Q445=+50                ;CLEARANCE HEIGHT ~
    Q592=+1                 ;TYPE OF DIMENSION
```

Machining variants

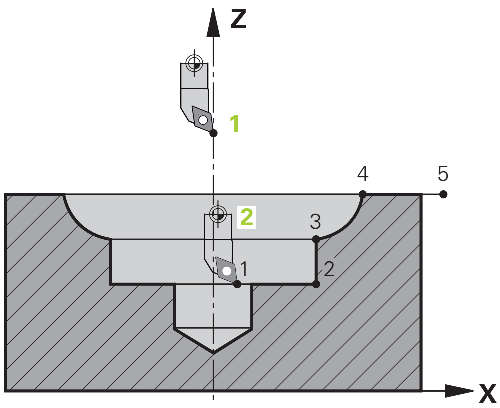
Before using Cycle **292**, you first need to define the desired turning contour in a subprogram and refer to this contour with Cycle **14** or **SEL CONTOUR**. Describe the turning contour on the cross section of a rotationally symmetrical body. Depending on the tool axis, use the following coordinates to define the turning contour:

Tool axis used	Axial coordinate	Radial coordinate
Z	Z	X
X	X	Y
Y	Y	Z

Example: If you are using the tool axis Z, program the turning contour in the axial direction in Z and the radius or diameter of the contour in X.

You can use this cycle for inside and outside machining. Some of the notes given in chapter "Notes", Page 696 are illustrated in the following. You will also find an example in "Example: Interpolation Turning Cycle 292", Page 742

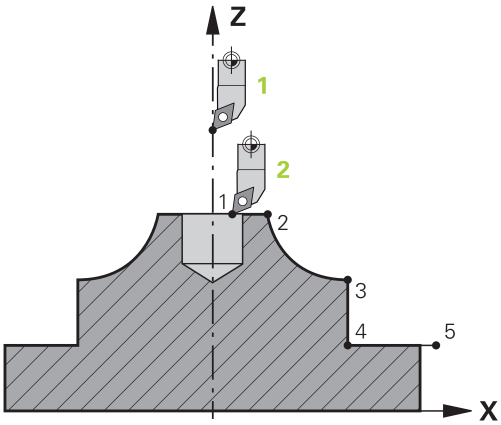
Inside machining



- The center of rotation is the position of the tool in the working plane when the cycle is called (1)
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while describing the contour! (2)
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control positions the tool to the contour starting point at rapid traverse in the tool axis direction. **Make sure that there is no material at the contour starting point.**

  You also need to take the following into account when programming the inside contour:
  - Program either monotonously increasing radial and axial coordinates (e.g., 1 to 5)
  - Or program monotonously decreasing radial and axial coordinates (e.g., 5 to 1)
  - Program inside contours with a radius greater than the tool radius.

Outside machining



- The center of rotation is the position of the tool in the working plane when the cycle is called (**1**)
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while describing the contour! (**2**)
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control positions the tool to the contour starting point at rapid traverse in the tool axis direction. **Make sure that there is no material at the contour starting point.**

  You also need to take the following into account when programming the outside contour:
  - Program either monotonously increasing radial coordinates and monotonously decreasing axial coordinates (e.g., 1 to 5)
  - Or program monotonously decreasing radial coordinates and monotonously increasing axial coordinates (e.g., 5 to 1)
  - Program outside contours with a radius greater than 0.

Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either mill (**Q560**=0) or turn (**Q560**=1) the contour. For each of the two machining modes, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

Spindle coupling off, Q560=0

Milling: Define the milling cutter in the tool table as usual by entering the length, radius, toroid cutter radius, etc.

Spindle coupling on, Q560=1

Turning: The geometry data of the turning tool are converted to the data of a milling cutter. You now have the following three possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

NOTICE

Danger of collision!

Collision may occur between the tool holder and workpiece during inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

- ▶ Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.
- **Define a turning tool in the turning tool table (toolturn.trn)**

  If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case, the orientation of the spindle to the center of rotation takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table) and parameter **Q336**.

  The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + **Q336**
Interpolation turning, inside	7	ORI + **Q336** + 180
Interpolation turning, outside	7	ORI + **Q336** + 180
Interpolation turning, inside	1	ORI + **Q336**
Interpolation turning, outside	8,9	ORI + **Q336**
Interpolation turning, inside	8,9	ORI + **Q336**

You can use the following tool types for interpolation turning:

- **TYPE**: **ROUGH**, with the machining directions **TO**: 1 or 7
- **TYPE**: **FINISH**, with the machining directions **TO**: 1 or 7
- **TYPE**: **BUTTON**, with the machining directions **TO**: 1 or 7

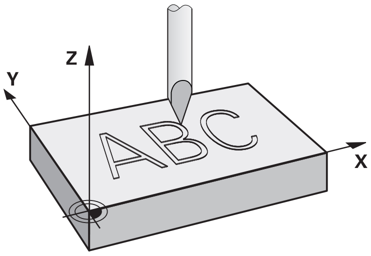
The following tool types cannot be used for interpolation turning:

- **TYPE**: **ROUGH**, with the machining directions **TO**: 2 to 6
- **TYPE**: **FINISH**, with the machining directions **TO**: 2 to 6
- **TYPE**: **BUTTON**, with the machining directions **TO**: 2 to 6
- **TYPE**: **RECESS**
- **TYPE**: **RECTURN**
- **TYPE**: **THREAD**

15.3.43 Cycle 225 ENGRAVING

ISO programming
G225

Application



This cycle is used to engrave texts on a flat surface of the workpiece. You can arrange the texts in a straight line or along an arc.

Cycle sequence

1. If the tool is beneath **Q204 2ND SET-UP CLEARANCE**, the control will first move to the value from **Q204**.
2. The control positions the tool in the working plane to the starting point of the first character.
3. The control engraves the text.
   - If **Q202 MAX. PLUNGING DEPTH** is greater than **Q201 DEPTH**, the control will engrave each character in a single infeed motion.
   - If **Q202 MAX. PLUNGING DEPTH** is less than **Q201 DEPTH**, the control will engrave each character in several infeed motions. The control will always complete the milling of a character before machining the next one.
4. After the control has engraved a character, it retracts the tool to the set-up clearance **Q200** above the workpiece surface.
5. The process steps 2 and 3 are repeated for all characters to be engraved.
6. Finally, the control retracts the tool to 2nd set-up clearance **Q204**.

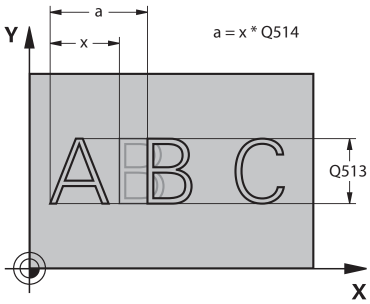
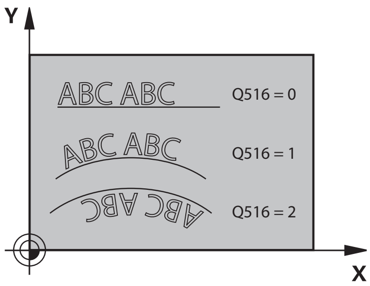
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Notes on programming

- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The text to be engraved can also be transferred with a string variable (**QS**).
- Parameter **Q347** influences the rotational position of the letters.
  If **Q374** = 0° to 180°, the characters are engraved from left to right.
  If **Q374** is greater than 180°, the direction of engraving is reversed.

Cycle parameters

Help graphic	Parameter
	QS500 Engraving text? Text to be engraved within quotation marks. Assignment of a string variable through the **Q** key of the numerical keypad. The **Q** key on the alphabetic keyboard represents normal text input. Input: Max. **255** characters
	**Q513 Character height?** Height of the characters to be engraved in mm Input: **0...999.999**
	Q514 Character spacing factor? The font used is called a proportional font. This means that the width of the characters varies depending on their shape. **X** = width of the character + default spacing. This factor allows you to influence the spacing. **Q514 = 0/1**: Default spacing between the characters **Q514 > 1**: The spacing between the characters is expanded. **Q514 < 1**: The spacing between the characters is reduced. This can lead to overlapping characters. Input: **0...10**
	Q515 Font? By default, the control uses the **DeJaVuSans** font.
	**Q516 Text on a line/on an arc(0-2)?** **0**: Engrave text in a straight line **1**: Engrave text along an arc **2**: Engrave text along the inside of a circular arc (circumferentially; not necessarily legible from below) Input: **0**, **1**, **2**
	Q374 Angle of rotation? Center angle if the text is arranged on an arc. Engraving angle when text is in a straight line. Input: **-360.000...+360.000**
	Q517 Radius of text on an arc? Radius of the arc in mm on which the control will engrave the text. 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**
	Q201 Depth? Distance between workpiece surface and engraving floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter

Q206 Feed rate for plunging?

Tool traversing speed in mm/min during plunging

Input: **0...99999.999** or **FAUTO**, **FU**

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q203 Workpiece surface coordinate?

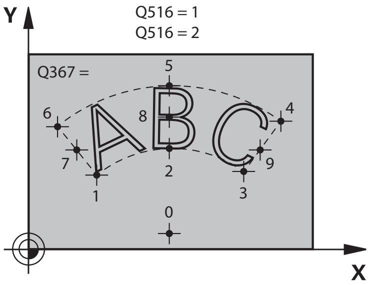
Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

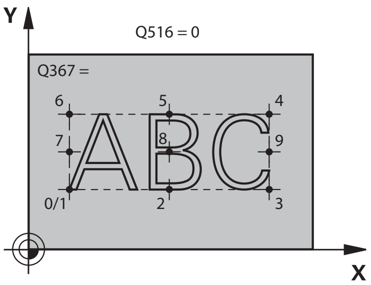
Input: **-99999.9999...+99999.9999**

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** or **PREDEF**





Q367 Reference for text position (0-6)?

Enter the reference for the position of the text here. Depending on whether the text will be engraved along a circular arc or in a straight line (parameter **Q516**), the following values can be entered:

Circle	Straight line
0 = Circle center	0 = Bottom left
1 = Bottom left	1 = Bottom left
2 = Bottom center	2 = Bottom center
3 = Bottom right	3 = Bottom right
4 = Top right	4 = Top right
5 = Top center	5 = Top center
6 = Top left	6 = Top left
7 = Center left	7 = Center left
8 = Center of text	8 = Center of text
9 = Center right	9 = Center right

Input: **0...9**

Help graphic	Parameter
	Q574 Maximum text length? Enter the maximum text length. The control also takes into account parameter **Q513** Character height. If **Q513 = 0**, the control engraves the text over exactly the length indicated in parameter **Q574**. The character height will be scaled accordingly. If **Q513 > 0**, the control checks whether the actual text length exceeds the maximum text length entered in **Q574**. If that is the case, the control displays an error message. Input: **0...999.999**
	Q202 Maximum plunging depth? Maximum infeed depth per cut. The machining operation is performed in several steps if this value is less than **Q201**. Input: **0...99999.9999**

Example

```
11 CYCL DEF 225 ENGRAVING ~
    QS500=""                ;ENGRAVING TEXT ~
    Q513=+10                ;CHARACTER HEIGHT ~
    Q514=+0                 ;SPACE FACTOR ~
    Q515=+0                 ;FONT ~
    Q516=+0                 ;TEXT ARRANGEMENT ~
    Q374=+0                 ;ANGLE OF ROTATION ~
    Q517=+50                ;CIRCLE RADIUS ~
    Q207=+500               ;FEED RATE MILLING ~
    Q201=-2                 ;DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q367=+0                 ;TEXT POSITION ~
    Q574=+0                 ;TEXT LENGTH ~
    Q202=+0                 ;MAX. PLUNGING DEPTH
```

Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers: **! # $ % & ' ( ) * + , - . / : ; < = > ? @ [\ ] _ ß CE**

> The control uses the special characters **%** and **** for special functions. If you want to engrave these characters, enter them twice in the text to be engraved (e.g., **%%**).

When engraving German umlauts, ß, ø, @, or the CE character, enter the character % before the character to be engraved:

Input	Algebraic sign
%ae	ä
%oe	ö
%ue	ü
%AE	Ä
%OE	Ö
%UE	Ü
%ss	ß
%D	ø
%at	@
%CE	CE

Characters that cannot be printed

Apart from text, you can also define certain non-printable characters for formatting purposes. Enter the special character **** before the non-printable characters.

The following formatting possibilities are available:

Input	Character
\n	Line break
\t	Horizontal tab (the tab width is permanently set to eight characters)
\v	Vertical tab (the tab width is permanently set to one line)

Engraving system variables

In addition to the standard characters, you can engrave the contents of certain system variables. Precede the system variable with **%**.

You can also engrave the current date, the current time, or the current calendar week. Do do so, enter **%time<x>**. **<x>** defines the format (e.g., 08 for DD.MM.YYYY.) (Identical to the **SYSSTR ID10321** function).

> Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9 (e.g., **%time08**).

Input	Character
%time00	DD.MM.YYYY hh:mm:ss
%time01	D.MM.YYYY h:mm:ss
%time02	D.MM.YYYY h:mm
%time03	D.MM.YY h:mm
%time04	YYYY-MM-DD hh:mm:ss
%time05	YYYY-MM-DD hh:mm
%time06	YYYY-MM-DD h:mm
%time07	YY-MM-DD h:mm
%time08	DD.MM.YYYY
%time09	D.MM.YYYY
%time10	D.MM.YY
%time11	YYYY-MM-DD
%time12	YY-MM-DD
%time13	hh:mm:ss
%time14	h:mm:ss
%time15	h:mm
%time99	ISO 8601 calendar week

> Properties:
> - 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.

Engraving the name and path of an NC program

Use Cycle **225** to engrave the name and path of an NC program.

Define Cycle **225** as usual. Precede the engraved text with **%**.

It is possible to engrave the name or path of an active or called NC program. For this purpose, define **%main<x>** or **%prog<x>**. (Identical to the **SYSSTR ID10010 NR1/2** function)

The following formatting possibilities are available:

Input	Meaning	Example
%main0	Full path of the active NC program	**TNC:\MILL.h**
%main1	Path to the directory of the active NC program	**TNC:**
%main2	Name of the active NC program	**MILL**
%main3	File type of the active NC program	**.H**
%prog0	Full path of the called NC program	**TNC:\HOUSE.h**
%prog1	Path to the directory of the called NC program	**TNC:**
%prog2	Name of the called NC program	**HOUSE**
%prog3	File type of the active NC program	**.H**

Engraving the counter reading

Cycle **225** allows you to engrave the current counter reading (provided on the PGM tab of the **Status** work status).

To do so, program Cycle **225** as usual and enter the text to be engraved, for example: **%count2**

The number after **%count** indicates how many digits the control will engrave. The maximum is nine digits.

Example: If you program **%count9** in the cycle with a momentary counter reading of 3, the control will engrave the following: 000000003

Further information: "Defining counters with FUNCTION COUNT", Page 1409

Operating notes

- In Simulation, the control simulates only the counter reading that you have specified directly in the NC program. The counter reading from the program run is not taken into account.

15.3.44 Cycle 232 FACE MILLING

ISO programming
G232

Application

With Cycle **232**, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

Related topics

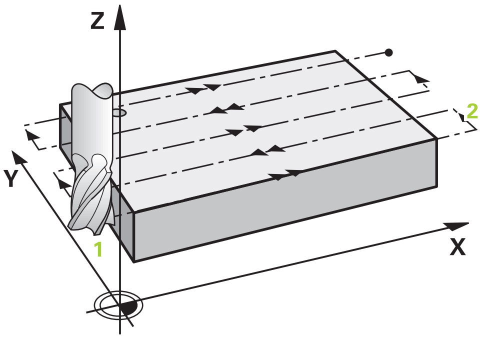
- Cycle **233 FACE MILLING**

  Further information: "Cycle 233 FACE MILLING ", Page 609

Cycle sequence

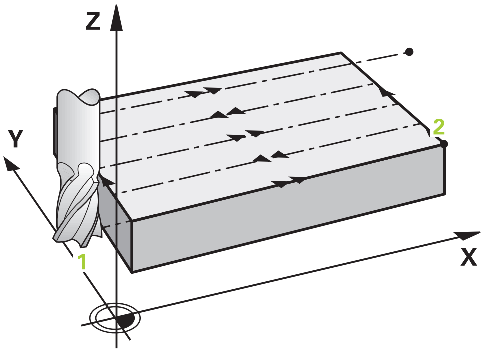
1. From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** using positioning logic: If the current position in the spindle axis is further away from the workpiece than the 2nd set-up clearance, the control positions the tool first in the working plane and then in the spindle axis. Otherwise, it first moves it to 2nd set-up clearance and then 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 tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

Strategy Q389=0



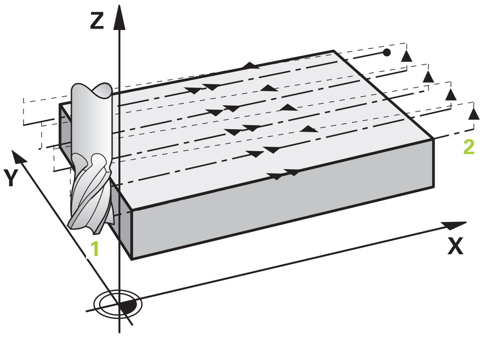
3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategy Q389=1



3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **at the edge** of the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.
4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**. The motion to the next pass again occurs at the edge of the workpiece.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been 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.

Strategy Q389=2



3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves it at the pre-positioning feed rate directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then returns to the current infeed depth and moves in the direction of end point **2**
6 The process is repeated until the programmed surface has been machined completely. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Notes

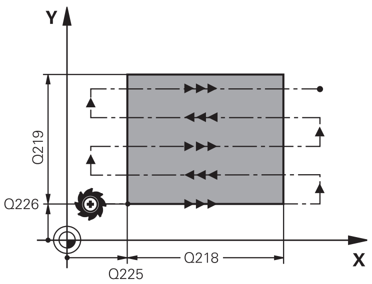
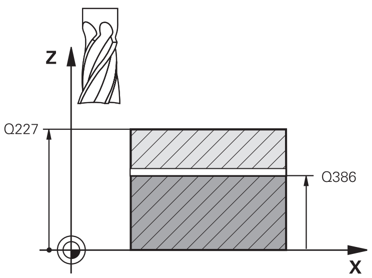
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

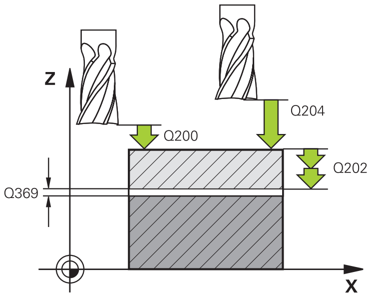
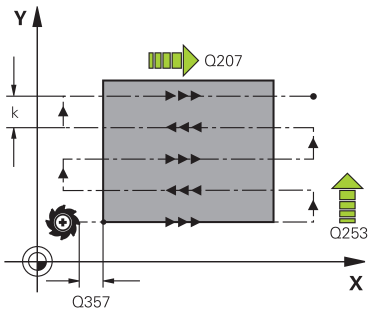
Notes on programming

- 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).
- Program **Q227** greater than **Q386**. The control will otherwise display an error message.

i	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
	Q389 Machining strategy (0/1/2)? Define how the control will machine 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**: Line-by-line machining, retraction and stepover at the positioning feed rate Input: **0**, **1**, **2**
	**Q225 Starting point in 1st axis?** Define the starting point coordinate of the surface to be machined 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? Define the starting point coordinate of the surface to be machined in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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**
	Q218 First side length? Length of the surface to be machined in the main axis of the working plane. Use the algebraic sign to specify the direction of the first milling path referenced to 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
	**Q202 Maximum plunging depth?** **Maximum** infeed per cut. The control calculates the actual plunging depth from the difference between the end point and starting point in the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Value used for the last infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q370 Max. 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. If you have entered a radius R2 in the tool table (e.g., cutter radius when using a face-milling cutter), the control reduces the stepover accordingly. Input: **0.001...1.999**
	**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**, **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy **Q389** = 2, the control moves the tool to set-up clearance above the current plunging depth to the starting point of the next pass. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

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**. 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** or **PREDEF**

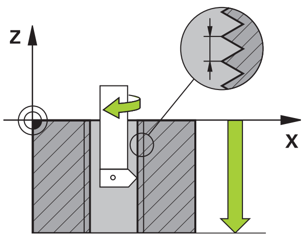
Example

```
11 CYCL DEF 232 FACE MILLING ~
    Q389=+2            ;STRATEGY ~
    Q225=+0            ;STARTNG PNT 1ST AXIS ~
    Q226=+0            ;STARTNG PNT 2ND AXIS ~
    Q227=+2.5          ;STARTNG PNT 3RD AXIS ~
    Q386=0             ;END POINT 3RD AXIS ~
    Q218=+150          ;FIRST SIDE LENGTH ~
    Q219=+75           ;2ND SIDE LENGTH ~
    Q202=+5            ;MAX. PLUNGING DEPTH ~
    Q369=+0            ;ALLOWANCE FOR FLOOR ~
    Q370=+1            ;MAX. OVERLAP ~
    Q207=+500          ;FEED RATE MILLING ~
    Q385=+500          ;FINISHING FEED RATE ~
    Q253=+750          ;F PRE-POSITIONING ~
    Q200=+2            ;SET-UP CLEARANCE ~
    Q357=+2            ;CLEARANCE TO SIDE ~
    Q204=+50           ;2ND SET-UP CLEARANCE
```

15.3.45 Cycle 18 THREAD CUTTING

ISO programming
G86

Application



Cycle **18 THREAD CUTTING** moves the tool with servo-controlled spindle from the momentary position with active speed to the specified depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.

Related topics

- Cycles for thread machining
  Further information: "Cycle 206 TAPPING ", Page 534
  Further information: "Cycle 207 RIGID TAPPING ", Page 537
  Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 540

Notes

NOTICE

Danger of collision!

If you do not program a pre-positioning step before programming the call of Cycle **18**, a collision might occur. Cycle **18** does not perform any approach or departure movements.

- Pre-position the tool before the start of the cycle.
- The tool moves from the current position to the entered depth after the cycle is called

NOTICE

Danger of collision!

If the spindle was switched on before the start of this cycle, Cycle **18** will switch it off and the cycle will execute with a stationary spindle! At the end, Cycle **18** will switch the spindle on again if it was on before the start of the cycle.

- Before starting this cycle, be sure to program a spindle stop! (For example with **M5**)
- At the end of Cycle **18**, the control restores the spindle to its state at cycle start. This means that if the spindle was switched off before this cycle, the control will switch it off again at the end of Cycle **18**.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

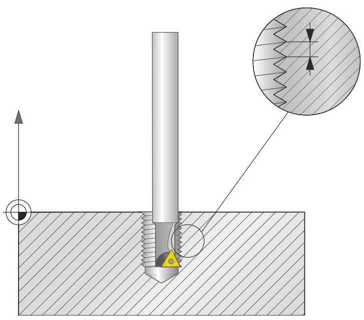
Notes on programming

- Before calling this cycle, program a spindle stop (for example with M5). The control automatically activates spindle rotation at the start of the cycle and deactivates it at the end.
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.

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

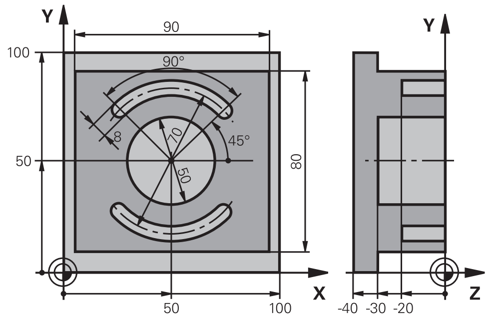
Help graphic	Parameter
	**Total hole depth?** Enter the thread depth relative to the current position. This value has an incremental effect. Input: **-999999999...+999999999**
	Thread pitch? Enter the thread pitch. The algebraic sign entered here differentiates between right-hand and left-hand threads: **+** = Right-hand thread (M3 with negative hole depth) **-** = Left-hand thread (M4 with negative hole depth) Input: **-99.9999...+99.9999**

Example

```
11 CYCL DEF 18.0 THREAD CUTTING
12 CYCL DEF 18.1 DEPTH-20
13 CYCL DEF 18.2 PITCH+1
```

15.3.46 Programming examples

Example: Milling pockets, studs and slots

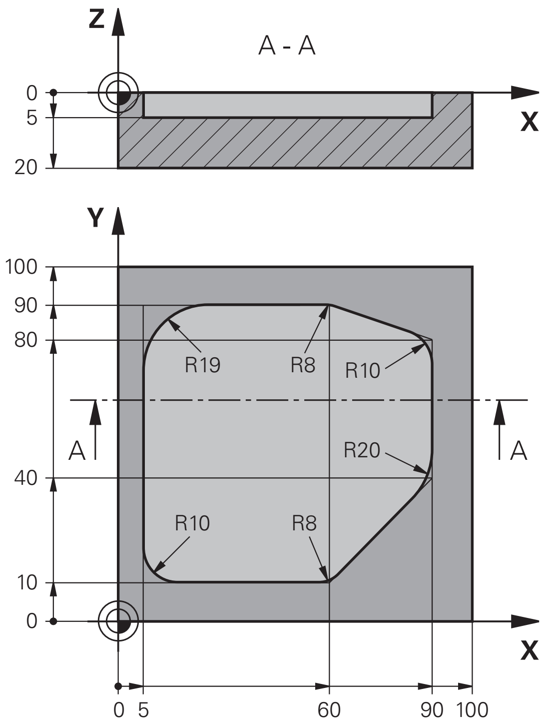


0 BEGIN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 6 Z S3500	; Tool call: roughing/finishing
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+90 ;FIRST SIDE LENGTH ~	
Q424=+100 ;WORKPC. BLANK SIDE 1 ~	
Q219=+80 ;2ND SIDE LENGTH ~	
Q425=+100 ;WORKPC. BLANK SIDE 2 ~	
Q220=+0 ;CORNER RADIUS ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q224=+0 ;ANGLE OF ROTATION ~	
Q367=+0 ;STUD POSITION ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-30 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+20 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q437=+0 ;APPROACH POSITION ~	
Q215=+0 ;MACHINING OPERATION ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q338=+10 ;INFEED FOR FINISHING ~	
Q385=+500 ;FINISHING FEED RATE	
6 L X+50 Y+50 R0 FMAX M99	; Cycle call for outside machining
7 CYCL DEF 252 CIRCULAR POCKET ~	
Q215=+0 ;MACHINING OPERATION ~	

Q223=+50 ;CIRCLE DIAMETER ~	
Q368=+0.2 ;ALLOWANCE FOR SIDE ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-30 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q366=+1 ;PLUNGE ~	
Q385=+750 ;FINISHING FEED RATE ~	
Q439=+0 ;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99	; Cycle call for circular pocket
9 TOOL CALL 3 Z S5000	; Tool call: slot milling cutter
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 254 CIRCULAR SLOT ~	
Q215=+0 ;MACHINING OPERATION ~	
Q219=+8 ;SLOT WIDTH ~	
Q368=+0.2 ;ALLOWANCE FOR SIDE ~	
Q375=+70 ;PITCH CIRCLE DIAMETR ~	
Q367=+0 ;REF. SLOT POSITION ~	
Q216=+50 ;CENTER IN 1ST AXIS ~	
Q217=+50 ;CENTER IN 2ND AXIS ~	
Q376=+45 ;STARTING ANGLE ~	
Q248=+90 ;ANGULAR LENGTH ~	
Q378=+180 ;STEPPING ANGLE ~	
Q377=+2 ;NR OF REPETITIONS ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-20 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q366=+2 ;PLUNGE ~	
Q385=+500 ;FINISHING FEED RATE ~	

Q439=+0 ;FEED RATE REFERENCE	
12 CYCL CALL	; Cycle call for slots
13 L Z+100 R0 FMAX	; Retract the tool, end program
14 M30	
15 END PGM C210 MM	

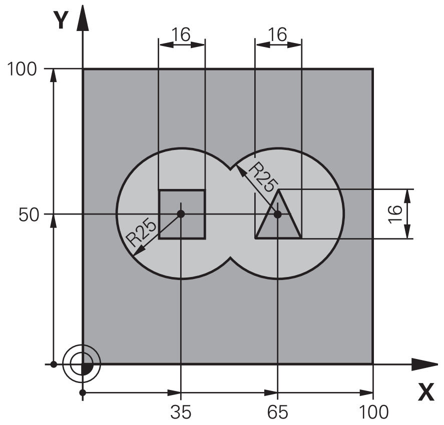
Example: Roughing-out and fine-roughing a pocket with SL Cycles



0 BEGIN PGM 1078634 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 15 Z S4500	; Tool call: coarse roughing tool (diameter: 30)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-5 ;MILLING DEPTH ~	
Q2=+1 ;TOOL PATH OVERLAP ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q4=+0 ;ALLOWANCE FOR FLOOR ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q7=+50 ;CLEARANCE HEIGHT ~	
Q8=+0.2 ;ROUNDING RADIUS ~	
Q9=+1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q12=+500 ;FEED RATE F. ROUGHNG ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q19=+200 ;FEED RATE FOR RECIP. ~	

Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+90 ;FEED RATE FACTOR ~	
Q404=+1 ;FINE ROUGH STRATEGY	
9 CYCL CALL	; Cycle call: coarse roughing
10 L Z+200 R0 FMAX	; Retract the tool
11 TOOL CALL 4 Z S3000	; Tool call: fine roughing tool (diameter: 8)
12 L Z+100 R0 FMAX M3	
13 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q12=+500 ;FEED RATE F. ROUGHNG ~	
Q18=+15 ;COARSE ROUGHING TOOL ~	
Q19=+200 ;FEED RATE FOR RECIP. ~	
Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+90 ;FEED RATE FACTOR ~	
Q404=+1 ;FINE ROUGH STRATEGY	
14 CYCL CALL	; Cycle call: fine roughing
15 L Z+200 R0 FMAX	; Retract the tool
16 M30	; End of program
17 LBL 1	; Contour subprogram
18 L X+5 Y+50 RR	
19 L Y+90	
20 RND R19	
21 L X+60	
22 RND R8	
23 L X+90 Y+80	
24 RND R10	
25 L Y+40	
26 RND R20	
27 L X+60 Y+10	
28 RND R8	
29 L X+5	
30 RND R10	
31 L X+5 Y+50	
32 LBL 0	
33 END PGM 1078634 MM	

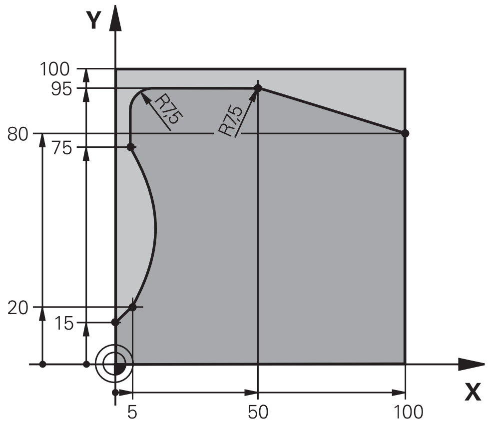
Example: Pilot drilling, roughing and finishing overlapping contours with SL Cycles



0 BEGIN PGM 2 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 TOOL CALL 204 Z S2500	; Tool call: drill (diameter: 12)
4 L Z+250 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1 /2 /3 /4	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-20 ;MILLING DEPTH ~	
Q2=+1 ;TOOL PATH OVERLAP ~	
Q3=+0.5 ;ALLOWANCE FOR SIDE ~	
Q4=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q7=+100 ;CLEARANCE HEIGHT ~	
Q8=+0.1 ;ROUNDING RADIUS ~	
Q9=-1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 21 PILOT DRILLING ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q13=+0 ;ROUGH-OUT TOOL	
9 CYCL CALL	; Cycle call: pilot drilling
10 L Z+100 R0 FMAX	; Retract the tool
11 TOOL CALL 6 Z S3000	; Tool call: roughing/finishing (D12)
12 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+350 ;FEED RATE F. ROUGHNG ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q19=+150 ;FEED RATE FOR RECIP. ~	

Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+100 ;FEED RATE FACTOR ~	
Q404=+0 ;FINE ROUGH STRATEGY	
13 CYCL CALL	; Cycle call: rough-out
14 CYCL DEF 23 FLOOR FINISHING ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+200 ;FEED RATE F. ROUGHNG ~	
Q208=+99999 ;RETRACTION FEED RATE	
15 CYCL CALL	; Cycle call: floor finishing
16 CYCL DEF 24 SIDE FINISHING ~	
Q9=+1 ;ROTATIONAL DIRECTION ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+400 ;FEED RATE F. ROUGHNG ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=-1 ;ROUGH-OUT TOOL	
17 CYCL CALL	; Cycle call: side finishing
18 L Z+100 R0 FMAX	; Retract the tool
19 M30	; End of program
20 LBL 1	; Contour subprogram 1: left pocket
21 CC X+35 Y+50	
22 L X+10 Y+50 RR	
23 C X+10 DR-	
24 LBL 0	
25 LBL 2	; Contour subprogram 2: right pocket
26 CC X+65 Y+50	
27 L X+90 Y+50 RR	
28 C X+90 DR-	
29 LBL 0	
30 LBL 3	; Contour subprogram 3: left square island
31 L X+27 Y+50 RL	
32 L Y+58	
33 L X+43	
34 L Y+42	
35 L X+27	
36 LBL 0	
37 LBL 4	; Contour subprogram 4: right triangular island
38 L X+65 Y+42 RL	
39 L X+57	
40 L X+65 Y+58	
41 L X+73 Y+42	
42 LBL 0	
43 END PGM 2 MM	

Example: Contour train



0 BEGIN PGM 3 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 TOOL CALL 10 Z S2000	; Tool call (diameter: 20)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 25 CONTOUR TRAIN ~	
Q1=-20 ;MILLING DEPTH ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q7=+250 ;CLEARANCE HEIGHT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+200 ;FEED RATE F. ROUGHNG ~	
Q15=+1 ;CLIMB OR UP-CUT ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q446=+0.01 ;RESIDUAL MATERIAL ~	
Q447=+10 ;CONNECTION DISTANCE ~	
Q448=+2 ;PATH EXTENSION	
8 CYCL CALL	; Cycle call
9 L Z+250 R0 FMAX	; Retract the tool, end program
10 M30	
11 LBL 1	; Contour subprogram
12 L X+0 Y+15 RL	
13 L X+5 Y+20	
13 CT X+5 Y+75	
14 CT X+5 Y+75	
15 L Y+95	
16 RND R7.5	
17 L X+50	

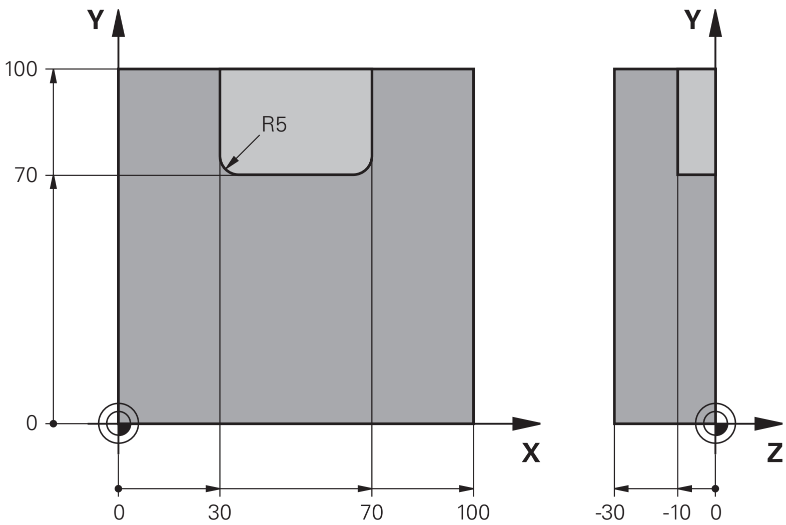
18 RND R7.5	
19 L X+100 Y+80	
20 LBL 0	
21 END PGM 3 MM	

Example: Open pocket and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will program an open pocket that is defined by means of an island and a boundary. Machining includes roughing and finishing of an open pocket.

Program sequence

- Tool call: Roughing cutter (Ø 20 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Roughing cutter (Ø 8 mm)
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 6 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**



0 BEGIN PGM OCM_POCKET MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S8000 F1500	; Tool call (diameter: 20 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 2	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-10 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+1 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	

Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+6500 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;INFEED STRATEGY	
8 CYCL CALL	; Cycle call
9 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6000 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+10 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;INFEED STRATEGY	
12 CYCL CALL	; Cycle call
13 TOOL CALL 23 Z S10000 F2000	; Tool call (diameter: 6 mm)
14 L Z+100 R0 FMAX M3	
15 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
16 CYCL CALL	; Cycle call
17 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0 ;INFEED FOR FINISHING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	

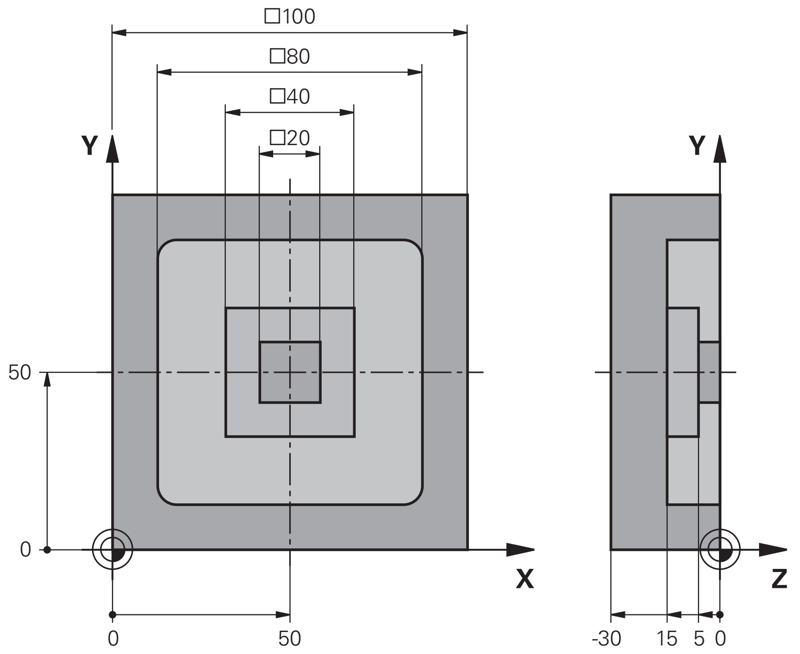
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
18 CYCL CALL	; Cycle call
19 M30	; End of program
20 LBL 1	; Contour subprogram 1
21 L X+0 Y+0	
22 L X+100	
23 L Y+100	
24 L X+0	
25 L Y+0	
26 LBL 0	
27 LBL 2	; Contour subprogram 2
28 L X+0 Y+0	
29 L X+100	
30 L Y+100	
31 L X+70	
32 L Y+70	
33 RND R5	
34 L X+30	
35 RND R5	
36 L Y+100	
37 L X+0	
38 L Y+0	
39 LBL 0	
40 END PGM OCM_POCKET MM	

Example: Program various depths with OCM cycles

The following NC program illustrates the use of OCM cycles. You will define one pocket and two islands at different heights. Machining includes roughing and finishing of a contour.

Program sequence

- Tool call: Roughing cutter (Ø 10 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 6 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**



0 BEGIN PGM OCM_DEPTH MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-30	
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL 5 Z S8000 F1500	; Tool call (diameter: 10 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 I3 = LBL 3 DEPTH5	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-15 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+0 ;OPEN BOUNDARY	

7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 CYCL CALL	; Cycle call
9 TOOL CALL 23 Z S10000 F2000	; Tool call (diameter: 6 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 CYCL CALL	; Cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0 ;INFEED FOR FINISHING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=+5 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
14 CYCL CALL	; Cycle call
15 M30	; End of program
16 LBL 1	; Contour subprogram 1
17 L X-40 Y-40	
18 L X+40	
19 L Y+40	
20 L X-40	
21 L Y-40	
22 LBL 0	

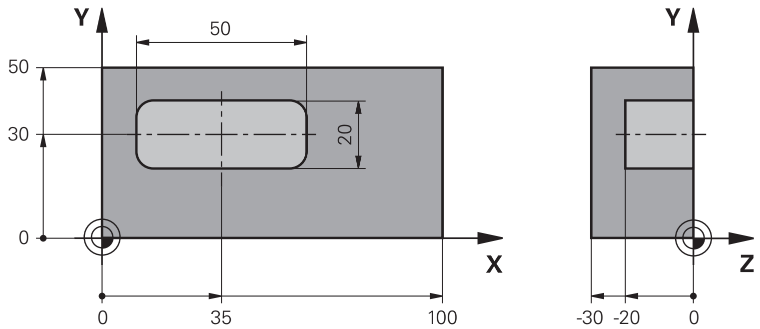
23 LBL 2	; Contour subprogram 2
24 L X-10 Y-10	
25 L X+10	
26 L Y+10	
27 L X-10	
28 L Y-10	
29 LBL 0	
30 LBL 3	; Contour subprogram 3
31 L X-20 Y-20	
32 L X+20	
33 L Y+20	
34 L X-20	
35 L Y-20	
36 LBL 0	
37 END PGM OCM_DEPTH MM	

Example: Face milling and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will face-mill a surface which will be defined by means of a boundary and an island. In addition, you will mill a pocket that contains an allowance for a smaller roughing tool.

Program sequence

- Tool call: Roughing cutter (Ø 12 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Roughing cutter (Ø 8 mm)
- Define Cycle **272** and call it again



0 BEGIN PGM FACE_MILL MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+50 Z+2	
3 TOOL CALL 6 Z S5000 F3000	; Tool call (diameter: 12 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 1 DEPTH2 P3 = LBL 2	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+2 ;SURFACE COORDINATE ~	
Q201=-22 ;DEPTH ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q369=+0 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+1 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+24 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+8000 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	

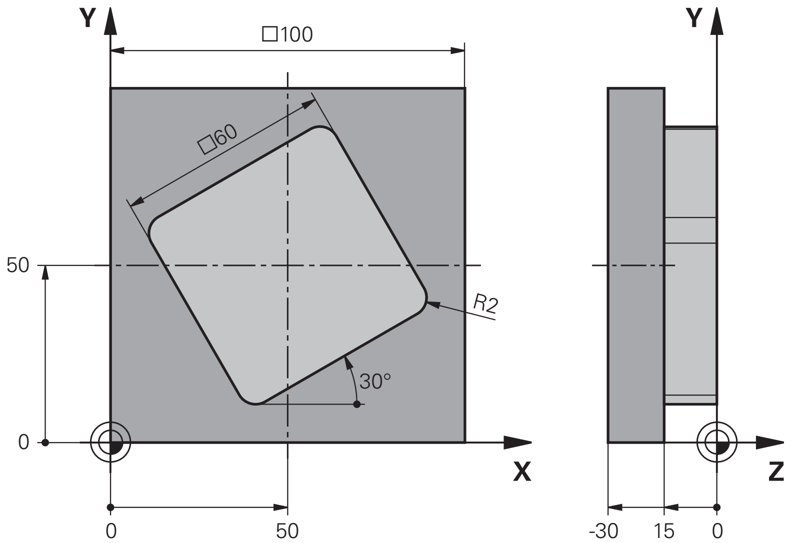
Q576=+8000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 L X+0 Y+0 R0 FMAX M99	; Cycle call
9 TOOL CALL 4 Z S6000 F4000	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+25 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+6 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
12 L X+0 Y+0 R0 FMAX M99	; Cycle call
13 M30	; End of program
14 LBL 1	; Contour subprogram 1
15 L X+0 Y+0	
16 L Y+50	
17 L X+100	
18 L Y+0	
19 L X+0	
20 LBL 0	
21 LBL 2	; Contour subprogram 2
22 L X+10 Y+30	
23 L Y+40	
24 RND R5	
25 L X+60	
26 RND R5	
27 L Y+20	
28 RND R5	
29 L X+10	
30 RND R5	
31 L Y+30	
32 LBL 0	
33 END PGM FACE_MILL MM	

Example: Contour with OCM figure cycles

The following NC program illustrates the use of OCM cycles. Machining includes roughing and finishing of a island.

Program sequence

- Tool call: Roughing cutter (Ø 8 mm)
- Define Cycle **1271**
- Define Cycle **1281**
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 8 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**

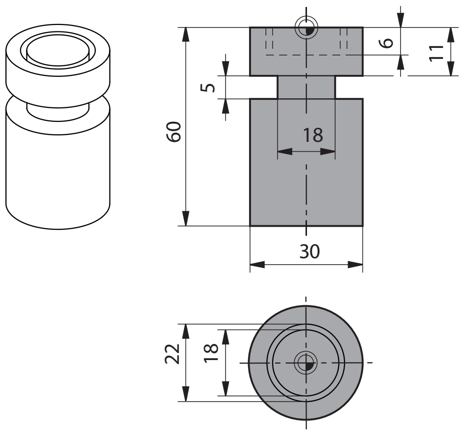


0 BEGIN PGM OCM_FIGURE MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 1271 OCM RECTANGLE ~	
Q650=+1 ;FIGURE TYPE ~	
Q218=+60 ;FIRST SIDE LENGTH ~	
Q219=+60 ;2ND SIDE LENGTH ~	
Q660=+0 ;CORNER TYPE ~	
Q220=+2 ;CORNER RADIUS ~	
Q367=+0 ;POCKET POSITION ~	
Q224=+30 ;ANGLE OF ROTATION ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-10 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR	

6 CYCL DEF 1281 OCM RECTANGLE BOUNDARY ~	
Q651=+100 ;LENGTH 1 ~	
Q652=+100 ;LENGTH 2 ~	
Q654=+0 ;POSITION REFERENCE ~	
Q655=+0 ;SHIFT 1 ~	
Q656=+0 ;SHIFT 2	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6800 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
9 TOOL CALL 24 Z S10000 F2000	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+4 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+15 ;INFEED FOR FINISHING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=+4 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
14 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
15 M30	; End of program
16 END PGM OCM_FIGURE MM	

Example: Interpolation turning with Cycle 291

The following NC program illustrates the use of Cycle **291 COUPLG.TURNG.INTERP.** This programming example shows how to machine an axial recess and a radial recess.



Tools

- Turning tool as defined in toolturn.trn: Tool no. 10: TO:1, ORI:0, TYPE:ROUGH, tool for axial recesses
- Turning tool as defined in toolturn.trn: Tool no. 11: TO:8, ORI:0, TYPE:ROUGH, tool for radial recesses

Program sequence

- Tool call: Tool for axial recess
- Start of interpolation turning: Description and call of Cycle **291**; **Q560** = 1
- End of interpolation turning: Description and call of Cycle **291**; **Q560** = 0
- Tool call: Recessing tool for radial recess
- Start of interpolation turning: Description and call of Cycle **291**; **Q560** = 1
- End of interpolation turning: Description and call of Cycle **291**; **Q560** = 0

> By converting parameter **Q561**, the turning tool is displayed in the simulation graphic as a milling tool.

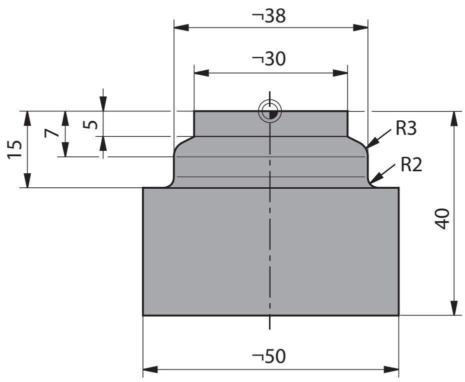
0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R15 L60	
2 TOOL CALL 10	; Tool call: tool for axial recess
3 CC X+0 Y+0	
4 LP PR+30 PA+0 R0 FMAX	; Retract the tool
5 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+1 ;CONVERT FROM TURNING TOOL	
6 CYCL CALL	; Call the cycle
7 LP PR+9 PA+0 RR FMAX	; Position the tool in the working plane
8 L Z+10 FMAX	
9 L Z+0.2 F2000	; Position the tool in the spindle axis

10 LBL 1	; Recessing on level surface (infeed: 0.2 mm, depth: 6 mm)
11 CP IPA+360 IZ-0.2 DR+ F10000	
12 CALL LBL 1 REP30	
13 LBL 2	; Retract from recess (step: 0.4 mm)
14 CP IPA+360 IZ+0.4 DR+	
15 CALL LBL 2 REP15	
16 L Z+200 R0 FMAX	; Retract to clearance height, deactivate radius compensation
17 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+0 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+0 ;CONVERT FROM TURNING TOOL	
18 CYCL CALL	; Call the cycle
19 TOOL CALL 11	; Tool call: tool for radial recess
20 CC X+0 Y+0	
21 LP PR+25 PA+0 R0 FMAX	; Retract the tool
22 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+1 ;CONVERT FROM TURNING TOOL	
23 CYCL CALL	; Call the cycle
24 LP PR+15 PA+0 RR FMAX	; Position the tool in the working plane
25 L Z+10 FMAX	
26 L Z-11 F7000	; Position the tool in the spindle axis
27 LBL 3	; Recessing on lateral surface (infeed: 0.2 mm, depth: 6 mm)
28 CC X+0.1 Y+0	
29 CP IPA+180 DR+ F10000	
30 CC X-0.1 Y+0	
31 CP IPA+180 DR+	
32 CALL LBL 3 REP15	
33 LBL 4	; Retract from recess (step: 0.4 mm)
34 CC X-0.2 Y+0	
35 CP PA+180 DR+	
36 CC X+0.2 Y+0	
37 CP IPA+180 DR+	
38 CALL LBL 4 REP8	
39 LP PR+50 FMAX	

40 L Z+200 R0 FMAX	; Retract to clearance height, deactivate radius compensation
41 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+0 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+0 ;CONVERT FROM TURNING TOOL	
42 CYCL CALL	; Call the cycle
43 TOOL CALL 11	; Repeated **TOOL CALL** in order to reset the conversion of parameter Q561
44 M30	
45 END PGM 5 MM	

Example: Interpolation Turning Cycle 292

The following NC program illustrates the use of Cycle **292 CONTOUR.TURNG.INTRP.** This programming example shows how to machine an outside contour with the milling spindle rotating.



Program sequence

- Tool call: Milling cutter D20
- Cycle **32 TOLERANCE**
- Reference to the contour with Cycle **14**
- Cycle **292 CONTOUR.TURNG.INTRP.**

0 BEGIN PGM 6 MM	
1 BLK FORM CYLINDER Z R25 L40	
2 TOOL CALL 10 Z S111	; Tool call: end mill D20
*** - ...**	; Use Cycle 32 to define the tolerance
3 CYCL DEF 32.0 TOLERANZ	
4 CYCL DEF 32.1 T0.05	
5 CYCL DEF 32.2 HSC-MODE:1	
6 CYCL DEF 14.0 CONTOUR	
7 CYCL DEF 14.1 CONTOUR LABEL1	
8 CYCL DEF 292 CONTOUR.TURNG.INTRP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q546=+3 ;CHANGE TOOL DIRECTN. ~	
Q529=+0 ;MACHINING OPERATION ~	
Q221=+0 ;SURFACE OVERSIZE ~	
Q441=+1 ;INFEED ~	
Q449=+15000 ;FEED RATE ~	
Q491=+15 ;CONTOUR START RADIUS ~	
Q357=+2 ;CLEARANCE TO SIDE ~	
Q445=+50 ;CLEARANCE HEIGHT ~	
Q592=+1 ;TYPE OF DIMENSION	
9 L Z+50 R0 FMAX M3	; Pre-position in the tool axis, spindle ON
10 L X+0 Y+0 R0 FMAX M99	; Pre-position in the working plane to the center of rotation, call the cycle
11 M30	; End of program

12 LBL 1	; LBL1 contains the contour
13 L Z+2 X+15	
14 L Z-5	
15 L Z-7 X+19	
16 RND R3	
17 L Z-15	
18 RND R2	
19 L X+27	
20 LBL 0	
21 END PGM 6 MM	

15.4 Cycles for milling and turning

15.4.1 Overview

The control offers the following cycles for turning operations:

Special cycles

Cycle		Call	Further information
800	**ADJUST XZ SYSTEM** (option 50) ■ Moving the tool to a suitable position relative to the turning spindle	**DEF**-active	Page 748
801	**RESET ROTARY COORDINATE SYSTEM** (option 50) ■ Resetting of cycle **800**	**DEF**-active	Page 756
892	**CHECK UNBALANCE** (option 50) ■ Checking the unbalance of the turning spindle	**DEF**-active	Page 757

Longitudinal turning cycles

Cycle		Call	Further information
811	**SHOULDER, LONGITDNL.** (option 50) ■ Longitudinal turning of rectangular shoulders	**CALL**-active	Page 762
812	**SHOULDER, LONG. EXT.** (option 50) ■ Longitudinal turning of rectangular shoulders ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 766
813	**TURN PLUNGE CONTOUR LONGITUDINAL** (option 50) ■ Longitudinal turning of shoulders with plunging elements	**CALL**-active	Page 771
814	**TURN PLUNGE LONGITUDINAL EXT.** (option 50) ■ Longitudinal turning of shoulders with plunging elements ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 775
810	**TURN CONTOUR LONG.** (option 50) ■ Longitudinal turning of turning contours of any shape ■ Removing stock paraxially	**CALL**-active	Page 780
815	**CONTOUR-PAR. TURNING** (option 50) ■ Longitudinal turning of turning contours of any shape ■ Removing of stock is performed parallel to the contour	**CALL**-active	Page 785

Face turning cycles

Cycle		Call	Further information
821	**SHOULDER, FACE** (option 50) ■ Face turning of rectangular shoulders	**CALL-**active	Page 789
822	**SHOULDER, FACE. EXT.** (option 50) ■ Face turning of rectangular shoulders ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL-**active	Page 793
823	**TURN TRANSVERSE PLUNGE** (option 50) ■ Face turning of shoulders with plunging elements	**CALL-**active	Page 798
824	**TURN PLUNGE TRANSVERSE EXT.** (option 50) ■ Face turning of shoulders with plunging elements ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL-**active	Page 802
820	**TURN CONTOUR TRANSV.** (option 50) ■ Face turning of turning contours of any shape	**CALL-**active	Page 807

Recess-turning cycles

Cycle		Call	Further information
841	**SIMPLE REC. TURNG., RADIAL DIR.** (option 50) ■ Recess turning of rectangular slots in longitudinal direction	**CALL-**active	Page 812
842	**ENH.REC.TURNNG, RAD.** (option 50) ■ Recess turning of slots in longitudinal direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL-**active	Page 816
851	**SIMPLE REC TURNG, AX** (option 50) ■ Recess turning of slots in transverse direction	**CALL-**active	Page 821
852	**ENH.REC.TURNING, AX.** (option 50) ■ Recess turning of slots in transverse direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL-**active	Page 825
840	**RECESS TURNG, RADIAL** (option 50) ■ Recess turning of slots of any shape in longitudinal direction	**CALL-**active	Page 830
850	**RECESS TURNG, AXIAL** (option 50)	**CALL-**active	Page 835

Cycle		Call	Further information
	■ Recess turning of slots of any shape in transverse direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface		

Recessing cycles

Cycle		Call	Further information
861	**SIMPLE RECESS, RADL.** (option 50) ■ Radial recessing of rectangular slots	**CALL-** active	Page 840
862	**EXPND. RECESS, RADL.** (option 50) ■ Radial recessing of rectangular slots ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL-** active	Page 845
871	**SIMPLE RECESS, AXIAL** (option 50) ■ Axial recessing of rectangular slots	**CALL-** active	Page 851
872	**EXPND. RECESS, AXIAL** (option 50) ■ Axial recessing of rectangular slots ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL-** active	Page 856
860	**CONT. RECESS, RADIAL** (option 50) ■ Radial recessing of slots of any shape	**CALL-** active	Page 862
870	**CONT. RECESS, AXIAL** (option 50) ■ Axial recessing of slots of any shape	**CALL-** active	Page 867

Thread cutting cycles

Cycle		Call	Further information
831	**THREAD LONGITUDINAL** (option 50) ■ Longitudinal turning of threads	**CALL-** active	Page 872
832	**THREAD EXTENDED** (option 50) ■ Longitudinal or face turning of threads and tapered threads ■ Definition of an approach path and an idle travel path	**CALL-** active	Page 876
830	**THREAD CONTOUR-PARALLEL** (option 50) ■ Longitudinal or face turning of threads of any shape ■ Definition of an approach path and an idle travel path	**CALL-** active	Page 881

Extended turning cycles

Cycle		Call	Further information
882	**SIMULTANEOUS ROUGHING FOR TURNING** (option 50 & 158) ■ Roughing of complex contours with different angles of inclination	**CALL**-active	Page 887
883	**TURNING SIMULTANEOUS FINISHING** (option 50 & 158) ■ Finishing of complex contours with different angles of inclination	**CALL**-active	Page 893

15.4.2 Working with turning cycles

Working with turning cycles

In turning cycles, the control takes the cutting geometry (**TO, RS, P-ANGLE, T-ANGLE**) of the tool into account in order to prevent damage to the defined contour elements. If it is not possible to machine the entire contour with the active tool, the control will display a warning.

You can use the turning cycles both for inside and outside machining. Depending upon the specific cycle, the control detects the machining position (inside or outside machining) via the starting position or tool position when the cycle is called. In some cycles you can also enter the machining position directly in the cycle. After modifying the machining position, check the tool position and the direction of rotation.

If you program **M136** before a cycle, the control interprets feed rate values in the cycle in mm/rev.; without **M136** in mm/min.

If you execute turning cycles with inclined machining (**M144**), the angles of the tool with respect to the contour change. The control automatically takes these modifications into account and thus also monitors the machining in inclined state to prevent contour damage.

Some cycles machine contours that you have written in a subprogram. You can program these contours with Klartext contouring functions. Before calling the cycle, you must program the cycle **14 CONTOUR** to define the subprogram number.

The turning cycles 81x - 87x as well 880, 882, and 883 must be called with **CYCL CALL** or **M99**. Before programming a cycle call, be sure to program:

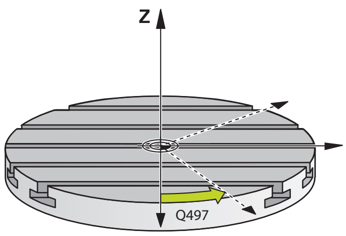
- Turning mode: **FUNCTION MODE TURN**
- Call a tool with **TOOL CALL**
- Direction of rotation of turning spindle (e.g., **M303**)
- Selection of speed or cutting speed: **FUNCTION TURNDATA SPIN**
- If you use feed rate per revolution mm/rev., **M136**
- Position the tool to a suitable starting point (e.g., **L X+130 Y+0 R0 FMAX**)
- Adapt the coordinate system, and align the tool: **CYCL DEF 800 ADJUST XZ SYSTEM**

15.4.3 Cycle 800 ADJUST XZ SYSTEM

ISO programming
G800

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.
The cycle is machine-dependent.



To be able to perform a turning operation, you need to position the tool appropriately relative to the turning spindle. For this purpose, you can use Cycle **800 ADJUST XZ SYSTEM**.

With turning operations, the inclination angle between the tool and turning spindle is important, for example to machine contours with undercuts. Cycle **800** provides various possibilities for aligning the coordinate system for an inclined machining operation:

- If you have positioned the tilting axis for inclined machining, you can use Cycle **800** to orient the coordinate system to the positions of the tilting axes (**Q530=0**). In this case, make sure to program **M144** or **M128**/**TCPM** for proper calculation of the orientation
- Cycle **800** calculates the required tilting axis angle based on the inclination angle **Q531** – depending on the strategy selected in the **INCLINED MACHINING Q530** parameter, the control positions the tilting axis with (**Q530=1**) or without compensating movement (**Q530=2**)
- Cycle **800** uses the inclination angle **Q531** to calculate the required tilting axis angle, but does not position the tilting axis (**Q530=3**). You need to position the tilting axis manually to the calculated values **Q120** (A axis), **Q121** (B axis), and **Q122** (C axis) after the cycle

If the milling spindle axis and the turning spindle axis are parallel to each other, you can use the **Precession angle Q497** to define any desired rotation of the coordinate system about the spindle axis (Z axis). This may be necessary if you have to bring the tool into a specific position due to space restrictions or if you want to improve your ability to observe a machining process. If the turning spindle and milling spindle axes are not parallel, only two precession angles are realistic for machining. The control selects the angle that is closest to the input value of **Q497**.

Cycle **800** positions the milling spindle such that the cutting edge is aligned relative to the turning contour. You can use a mirrored version of the tool (**REVERSE TOOL Q498**); this offsets the milling spindle by 180°. In this way, you can use your tools for both inside and outside machining. Position the cutting edge at the center of the turning spindle by using a positioning block, such as **L Y+0 R0 FMAX**.

- If you change the position of a tilting axis, you need to run Cycle **800** again to align the coordinate system.
- Check the orientation of the tool before machining.

Eccentric turning

Sometimes it is not possible to clamp a workpiece such that the axis of rotation is aligned with the axis of the turning spindle. For example, this is the case with large or rotationally non-symmetric workpieces. The eccentric turning **Q535** function in Cycle **800** enables you to perform turning in such cases as well.

During eccentric turning, more than one linear axis is coupled to the turning spindle. The control compensates the eccentricity by performing circular compensating movements with the coupled linear axes.

This function must be enabled and adapted by the machine manufacturer.

If you machine with high speed and a high amount of eccentricity, you need to program large feed rates for the linear axes in order to perform the movements synchronously. If these feed rates are not met, the contour would be damaged. The control therefore generates an error message if 80 % of a maximum axis speed or acceleration is exceeded. If this occurs, reduce the speed.

Operating information

NOTICE
Danger of collision! The control performs compensating movements during coupling and decoupling. There is a danger of collision! ▶ Coupling and decoupling must be performed while the spindle is stationary

NOTICE
Danger of collision! Collision monitoring (DCM) is not active during eccentric turning. The control displays a corresponding warning during eccentric turning. There is a danger of collision. ▶ Check the machining sequence by using the simulation

NOTICE
Caution: Danger to the tool and workpiece! The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life. ▶ Select the technology data in such a way that no vibrations (resonances) occur

- Turn a test cut before the actual machining operation to ensure that the required speeds can be attained.
- The linear axis positions resulting from the compensation are displayed by the control only in the ACTUAL value position display.

Effect

With Cycle **800 ADJUST XZ SYSTEM**, the control aligns the workpiece coordinate system and orients the tool correspondingly. Cycle **800** is effective until it is reset by Cycle **801**, or until Cycle **800** is redefined. Some cycle functions of Cycle **800** are implicitly reset by other factors:

- Mirroring of tool data (**Q498 REVERSE TOOL**) is reset by a tool call with **TOOL CALL**
- The **ECCENTRIC TURNING Q535** function is reset at the end of the program or if the program is aborted (internal stop)

Notes

The machine manufacturer configures your machine tool. If the tool spindle was defined as an axis in the kinematic model during this configuration, the feed-rate potentiometer is effective for movements related to Cycle **800**.

The machine manufacturer can configure a grid for the positioning of the tool spindle.

NOTICE

Danger of collision!

If the milling spindle was defined as an NC axis in turning mode, then the control is able to derive a tool reversal from the axis position. However, if the milling spindle was defined as a spindle, there is a risk that the tool reversal definition might get lost! There is a danger of collision!

- Enable tool reversal again after a **TOOL CALL** block

NOTICE

Danger of collision!

If **Q498** = 1 and you additionally program the **FUNCTION LIFTOFF ANGLE TCS** function, then there might be two different results, depending on the configuration. If the tool spindle has been defined as an axis, the **LIFTOFF** will be included in the rotation during tool reversal. If the tool spindle has been defined as a kinematic transformation, then the **LIFTOFF** will **not** be included in the rotation during tool reversal! There is a danger of collision!

- Carefully test the NC program or program section in **Single Block** mode of the **Program Run** operating mode
- If required, change the algebraic sign of the SPB angle.

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool must be clamped and measured in the correct position.
- Cycle **800** positions only the first rotary axis based on the tool position. If an **M138** is activated, then this limits the selection to the defined rotary axes. If you want to move other rotary axes to a specific position, then position theses axes correspondingly before running Cycle **800**.

  Further information: "Taking rotary axes into account during machining operations with M138", Page 1347

Notes on programming

- You can mirror the tool data (**Q498 REVERSE TOOL**) only if a turning tool has been selected.
- To reset Cycle **800**, program Cycle **801 RESET ROTARY COORDINATE SYSTEM**.
- Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine manufacturer) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.
- If the workpiece is to be rotated about the workpiece spindle, then use an offset of the workpiece spindle in the preset table. Basic rotations are not permitted; the control issues an error message.
- If you set parameter **Q530** "Inclined machining" to 0 (tilting axes must have been positioned previously), make sure to program **M144** or **TCPM/M128** beforehand.
- If, in parameter **Q530** “Inclined machining,” you use the settings 1: MOVE, 2: TURN and 3: STAY, then the control, depending on the machine configuration, activates function **M144** or TCPM

  Further information: "Turning (option 50)", Page 236

Cycle parameters

Help graphic	Parameters
	Q497 Precession angle? Angle at which the control positions the tool. Input: **0.0000...359.9999**
	Q498 Reverse tool (0=no/1=yes)? Mirror tool for inside/outside machining. Input: **0**, **1**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **0**: Maintain tilting axis position (axis must be positioned beforehand) **1**: Automatically position the tilting axis, and orient the tool tip (MOVE). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (TURN) **3**: Do not position the tilting axis. Position the tilting axes later in a separate positioning block (STAY). The control stores the position values in the parameters **Q120** (A axis), **Q121** (B axis) and **Q122** (C axis). Input: **0**, **1**, **2**, **3**
	Q531 Angle of incidence? Angle of incidence for positioning the tool Input: **-180...+180**
	Q532 Feed rate for positioning? Traversing speed of the tilting axis during automatic positioning Input: **0.001...99999.999**, or **FMAX**
	Q533 Preferred dir. of incid. angle? **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**

Help graphic	Parameters
	Q535 Eccentric turning? Couple the axes for the eccentric turning operation: **0**: Deactivate axis couplings **1**: Activate axis couplings. The center of rotation is located at the active preset **2**: Activate axis couplings. The center of rotation is located at the active datum **3**: Do not change the axis couplings Input: **0**, **1**, **2**, **3**
	Q536 Eccentric turning without stop? Interrupt program run before the axes are coupled: **0**: Stop before the axes are coupled again. In stopped condition, the control opens a window in which the amount of eccentricity and the maximum deflection of the individual axes are displayed. You can then continue the machining operating with **NC-Start** or select **ABBRUCH** **1**: Axes are coupled without stopping beforehand Input: **0**, **1**
	Q599 or **QS599 Retraction path/macro?** Retraction prior to execution of positioning movements in the rotary axis or tool axis: **0**: No retraction **-1**: Maximum retraction with **M140 MB MAX**, see "Retracting in the tool axis with M140", Page 1348 **> 0**: Path for the retraction in **mm** or **inches** **"..."**: Path for an NC program that will be called as a user macro. **Further information:** "User macro", Page 755 Input: **-1...9999** in the case of text entry: maximum **255** characters or **QS** parameter

Example

```
11 CYCL DEF 800 ADJUST XZ SYSTEM ~
    Q497=+0              ;PRECESSION ANGLE ~
    Q498=+0              ;REVERSE TOOL ~
    Q530=+0              ;INCLINED MACHINING ~
    Q531=+0              ;ANGLE OF INCIDENCE ~
    Q532=+750            ;FEED RATE ~
    Q533=+0              ;PREFERRED DIRECTION ~
    Q535=+3              ;ECCENTRIC TURNING ~
    Q536=+0              ;ECCENTRIC W/O STOP ~
    Q599=-1              ;RETRACT
```

User macro

The 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 **PGM CALL** function, 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: "Variables: Q, QL, QR and QS parameters", Page 1362

Example of a user macro for retraction

0 BEGIN PGM RET MM	
1 FUNCTION RESET TCPM	; Reset TCPM
2 L Z-1 R0 FMAX M91	; Traverse with M91
3 FN 10: IF +Q533 NE +0 GOTO LBL "DEF_DIRECTION"	; If Q533 (preferred direction from Cycle 800) is not equal to 0, then jump to LBL "DEF_DIRECTION"
4 FN 18: SYSREAD QL1 = ID240 NR1 IDX4	; Read system data (nominal position in the REF system) and store in QL1
5 QL0 = 500 * SGN QL1	; SGN = Check algebraic sign
6 FN 9: IF +0 EQU +0 GOTO LBL "MOVE"	; Jump to LBL MOVE
7 LBL "DIRECTION"	
8 QL0 = 500 * SGN Q533	; SGN = Check algebraic sign
9 LBL "MOVE"	
10 L X-500 Y+QL0 R0 FMAX M91	; Retraction with M91
11 END PGM RET MM	

15.4.4 Cycle 801 RESET ROTARY COORDINATE SYSTEM

ISO programming
G801

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer. The cycle is machine-dependent.

Cycle **801** resets the following settings you have programmed with Cycle **800**:

- Precession angle **Q497**
- Reverse tool **Q498**

If you executed the eccentric turning function with Cycle **800**, please note the following: Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine manufacturer) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.

i	Cycle **801** does not orient the tool to the starting position. If a tool was oriented with Cycle **800**, it remains in this position also after resetting.

Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- With Cycle **801 RESET ROTARY COORDINATE SYSTEM**, you can reset the settings you have made with Cycle **800 ADJUST XZ SYSTEM**.

Notes on programming

- To reset Cycle **800**, program Cycle **801 RESET ROTARY COORDINATE SYSTEM**.
- Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine manufacturer) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.

Cycle parameters

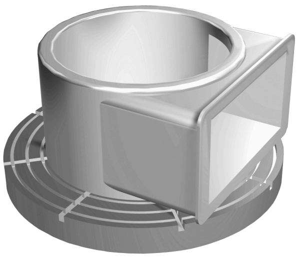
Help graphic	Parameter
	Cycle **801** does not have a cycle parameter. Close cycle input with the **END** key.

15.4.5 Cycle 892 CHECK UNBALANCE

ISO programming
G892

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



An unbalance can occur when turning an unsymmetrical workpiece, such as a pump body. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle **892 CHECK UNBALANCE**, the control checks the unbalance of the turning spindle. This cycle uses two parameters. **Q450** describes the maximum unbalance and **Q451** the maximum spindle speed. **If the maximum unbalance is exceeded, an error message is displayed and the NC program is aborted.** If the maximum unbalance is not exceeded, the control executes the NC program without interruption. This function protects the machine mechanics. It enables you to take action if an excessive unbalance is detected.

Notes

Your machine manufacturer configures Cycle **892**.

Your machine manufacturer defines the function of Cycle **892**.

The turning spindle rotates during the unbalance check.

This function can also be run on machines with more than one turning spindle. Contact the machine manufacturer for further information.

You need to check the applicability of the control's internal unbalance functionality for each of your machine types. If the unbalance amplitude of the turning spindle has very little effect on the adjoining axes, it might not be possible to calculate useful unbalance values from the determined results. In this case, you will have to use a system with external sensors for unbalance monitoring.

NOTICE

Danger of collision!

Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any imbalance. If high unbalance loads are not compensated for, then this may lead to defects on the machine.

- Before starting a new machining cycle, run Cycle **892**.
- If required, use balancing weights to compensate for any unbalance.

NOTICE

Danger of collision!

The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps. If high unbalance loads are not compensated, then this may lead to defects on the machine

- Make sure to also run Cycle **892** between the machining steps.
- If required, use balancing weights to compensate for any unbalance.

NOTICE

Danger of collision!

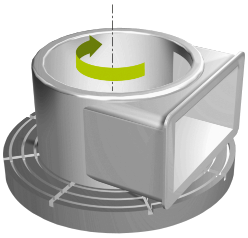
High unbalance loads, especially in combination with a high mass, may damage the machine. Consider the mass and unbalance of the workpiece when choosing the speed.

- Do not program high speeds with heavy workpieces or high unbalance loads.

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- If Cycle **892 CHECK UNBALANCE** has aborted the NC program, then we recommend that you use the manual MEASURE UNBALANCE cycle. With this cycle, the control determines the unbalance and calculates the mass and position of a balancing weight.

  Further information: "Unbalance in turning operations", Page 246

Cycle parameters

Help graphic	Parameter
	**Q450 Max. permissible runout?** Specifies the maximum runout of a sinusoidal unbalance signal in millimeters (mm). The signal results from the following error of the measuring axis and from the spindle revolutions. Input: **0...99999.9999**
	Q451 Rotational speed? Enter the rotational speed in revolutions per minute. The test for an unbalance begins with a low initial speed (e.g., 50 rpm). It is then automatically increased by specified increments (e.g., 25 rpm). until the maximum speed defined in parameter **Q451** is reached. Spindle speed override is disabled. Input: **0...99999**

Example

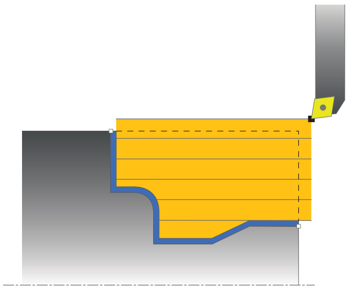
```
11 CYCL DEF 892 CHECK UNBALANCE ~
    Q450=+0                 ;MAXIMUM RUNOUT ~
    Q451=+50                ;SPEED
```

15.4.6 Fundamentals of turning cycles

Refer to your machine manual.

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

Option 50 must have been enabled.



The pre-positioning of the tool has a decisive influence on the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when the cycle is called. When calculating the area to be machined, the control takes into account the starting point and the end point defined in the cycle or of the contour defined in the cycle. If the starting point is within the area to be machined, then the control positions the tool at the set-up clearance beforehand in some cycles.

The direction of stock removal is longitudinal to the rotary axis for Cycles **81x** and transverse to the rotary axis for Cycles **82x**. In Cycle **815**, the movements are contour-parallel.

The cycles can be used for inside and outside machining. The control takes the information for this from the position of the tool or from the definition in the cycle.

Further information: "Working with turning cycles", Page 747

For cycles in which a defined contour is machined (Cycles **810**, **820**, and **815**), the direction set when programming the contour determines the machining direction.

In cycles for turning you can specify the machining strategies of roughing, finishing or complete machining.

NOTICE

Danger of collision!

The turning cycles position the tool automatically to the starting point during finishing. The approach strategy is influenced by the position of the tool when the cycle is called. The decisive factor is whether the tool is located inside or outside an envelope contour when the cycle is called. The envelope contour is the programmed contour, enlarged by the set-up clearance. If the tool is within the envelope contour, the cycle positions the tool at the defined feed rate directly to the starting position. This can cause contour damage.

- Position the tool at a sufficient distance from the starting point to prevent the possibility of contour damage
- If the tool is outside the envelope contour, positioning to the envelope contour is performed at rapid traverse, and at the programmed feed rate within the envelope contour.

> The control monitors the length of the cutting edge **CUTLENGTH** in the turning cycles. If the cutting depth programmed in the turning cycle is greater than the length of the cutting edge defined in the tool table, then the control issues a warning. In this case, the cutting depth will be reduced automatically in the machining cycle.

Execution with a FreeTurn tool

The control supports the execution of the contours with FreeTurn tools in the cycles **81x** and **82x**. This method allows you to perform the most common turning operation with just one tool. Thanks to the flexible tool, machining times can be reduced because the control does not need to change tools as much.

Requirements

- The tool must be correctly defined.

  Further information: "Turning operation with FreeTurn tools", Page 244

NOTICE

Danger of collision!

The shaft length of the turning tool limits the diameter that can be machined. There is a risk of collision during machining!

- ▶ Check the machining sequence in the simulation

> - The NC program remains unchanged except for the calling of the FreeTurn cutting edges.
>
>   **Further information:** "Example: Turning with a FreeTurn tool", Page 906
> - If you use a FreeTurn tool for machining, the control will internally switch the kinematics. This can lead to movements changing the positions of the cutting edge. In this case, the control will display a warning message.
>
>   If the control displays a warning message during simulation, HEIDENHAIN recommends that you run the program once without a workpiece. It is possible that the control does not display a warning during program run because the simulation does not show all movements, such as PLC positioning movements. The simulation may thus differ from the actual machining process.

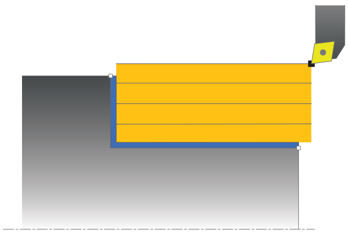
15.4.7 Cycle 811 SHOULDER, LONGITDNL.

ISO programming

G811

Application

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



This cycle enables you to carry out longitudinal turning of right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The cycle processes the area from the tool position to the end point defined in the cycle.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 The control performs a paraxial infeed movement at rapid traverse.
3 The control finishes the contour of the finished part at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

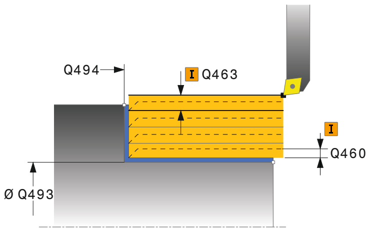
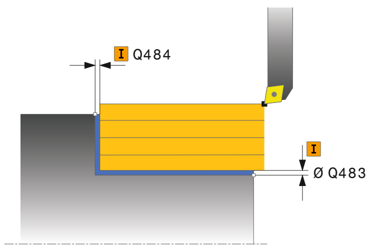
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

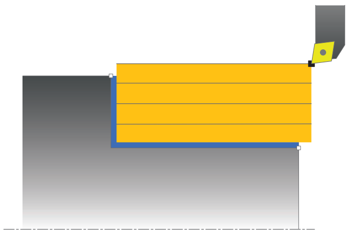
```
11 CYCL DEF 821 SHOULDER, LONGITDNL. ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q493=+50                 ;DIAMETER AT CONTOUR END ~
    Q494=-55                 ;CONTOUR END IN Z ~
    Q463=+3                  ;MAX. CUTTING DEPTH ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q506=+0                  ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 R0 FMAX M303
13 CYCL CALL
```

15.4.8 Cycle 812 SHOULDER, LONG. EXT.

ISO programming
G812

Application

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



This cycle enables you to run longitudinal turning of shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the X coordinate and then in the Z coordinate to set-up clearance and starts the cycle there.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the starting point lies in the area to be machined, the control positions the tool to set-up clearance beforehand.

1. The control performs a paraxial infeed movement at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

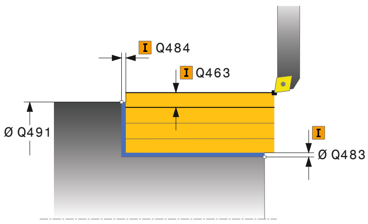
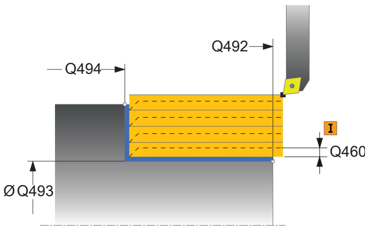
Notes

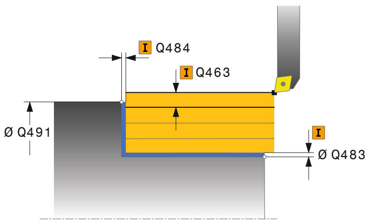
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
 	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
 	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

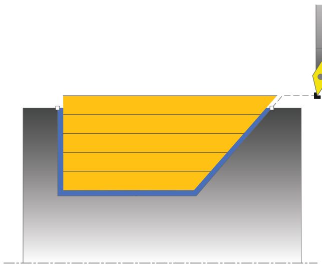
```
11 CYCL DEF 812 SHOULDER, LONG. EXT. ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q491=+75               ;DIAMETER AT CONTOUR START ~
    Q492=+0                ;CONTOUR START IN Z ~
    Q493=+50               ;DIAMETER AT CONTOUR END ~
    Q494=-55               ;CONTOUR END IN Z ~
    Q495=+5                ;ANGLE OF CIRCUM. SURFACE ~
    Q501=+1                ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5              ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5              ;RADIUS OF CONTOUR EDGE ~
    Q496=+0                ;ANGLE OF FACE ~
    Q503=+1                ;TYPE OF END ELEMENT ~
    Q504=+0.5              ;SIZE OF END ELEMENT ~
    Q463=+3                ;MAX. CUTTING DEPTH ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q506=+0                ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.9 Cycle 813 TURN PLUNGE CONTOUR LONGITUDINAL

ISO programming
G813

Application

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



This cycle enables you to run longitudinal turning of shoulders with plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

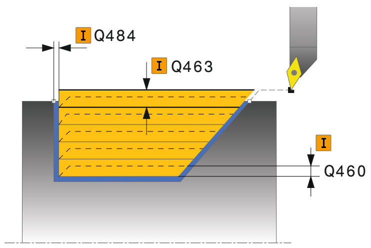
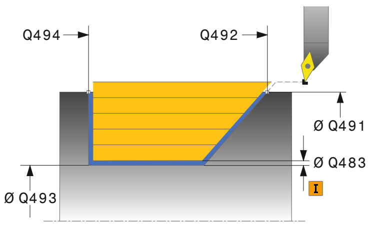
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point for the plunging path Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle of plunging flank. The reference angle is the line perpendicular to the rotary axis. Input: **0...89.9999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**

Help graphic	Parameter
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

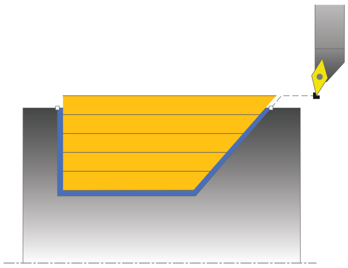
```
11 CYCL DEF 813 TURN PLUNGE CONTOUR LONGITUDINAL ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=-10                ;CONTOUR START IN Z ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-55                ;CONTOUR END IN Z ~
    Q495=+70                ;ANGLE OF SIDE ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 R0 FMAX M303
13 CYCL CALL
```

15.4.10 Cycle 814 TURN PLUNGE LONGITUDINAL EXT.

ISO programming
G814

Application

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



This cycle enables you to run longitudinal turning of shoulders with plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

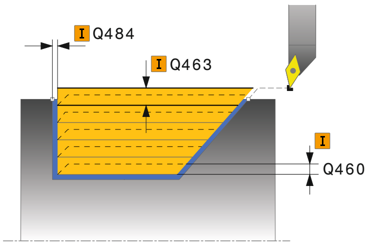
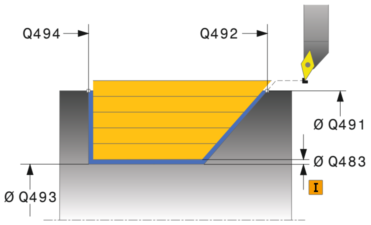
Notes

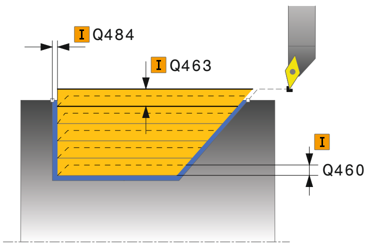
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
 	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point for the plunging path Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle of plunging flank. The reference angle is the line perpendicular to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

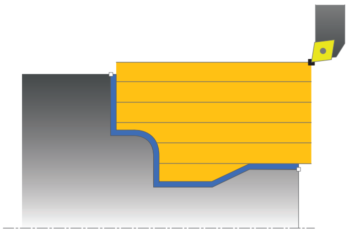
```
11 CYCL DEF 814 TURN PLUNGE LONGITUDINAL EXT. ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q491=+75                 ;DIAMETER AT CONTOUR START ~
    Q492=-10                 ;CONTOUR START IN Z ~
    Q493=+50                 ;DIAMETER AT CONTOUR END ~
    Q494=-55                 ;CONTOUR END IN Z ~
    Q495=+70                 ;ANGLE OF SIDE ~
    Q501=+1                  ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5                ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5                ;RADIUS OF CONTOUR EDGE ~
    Q496=+0                  ;ANGLE OF FACE ~
    Q503=+1                  ;TYPE OF END ELEMENT ~
    Q504=+0.5                ;SIZE OF END ELEMENT ~
    Q463=+3                  ;MAX. CUTTING DEPTH ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q506=+0                  ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.11 Cycle 810 TURN CONTOUR LONG.

ISO programming
G810

Application

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



This cycle enables you to run longitudinal turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction. The longitudinal cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

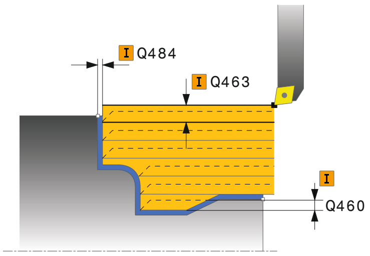
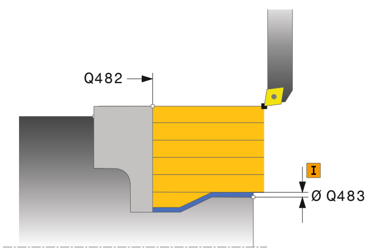
- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

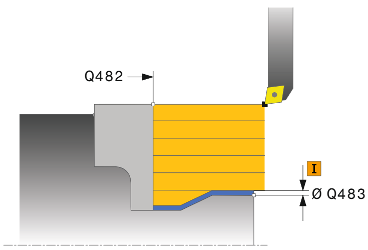
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q487 Allow plunging (0/1)? Permit the machining of plunging elements: **0**: Do not machine any plunging elements **1**: Machine plunging elements Input: **0**, **1**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	**Q482 Value of cutting limit in Z?** Z value for contour limit Input: **-99999.999...+99999.999**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

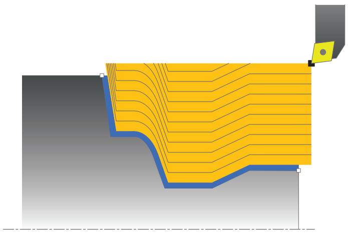
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 810 TURN CONTOUR LONG. ~
    Q215=+0                   ;MACHINING OPERATION ~
    Q460=+2                   ;SAFETY CLEARANCE ~
    Q499=+0                   ;REVERSE CONTOUR ~
    Q463=+3                   ;MAX. CUTTING DEPTH ~
    Q478=+0.3                 ;ROUGHING FEED RATE ~
    Q483=+0.4                 ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                 ;OVERSIZE IN Z ~
    Q505=+0.2                 ;FINISHING FEED RATE ~
    Q487=+1                   ;PLUNGE ~
    Q488=+0                   ;PLUNGING FEED RATE ~
    Q479=+0                   ;CONTOUR MACHINING LIMIT ~
    Q480=+0                   ;DIAMETER LIMIT VALUE ~
    Q482=+0                   ;LIMIT VALUE Z ~
    Q506=+0                   ;CONTOUR SMOOTHING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Z-35
22 RND R5
23 L  X+50  Z-40
24 L  Z-55
25 CC  X+60  Z-55
26 C  X+60  Z-60
27 L  X+100
28 LBL 0
```

15.4.12 Cycle 815 CONTOUR-PAR. TURNING

ISO programming
G815

Application

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



This cycle enables you to run turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning with roughing is contour-parallel.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and end point. The cut is performed in contour-parallel mode at the defined feed rate **Q478**.
3 The control returns the tool at the defined feed rate back to the starting position in the X coordinate.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

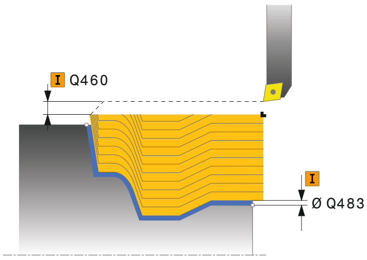
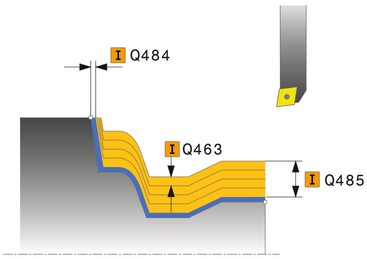
Notes

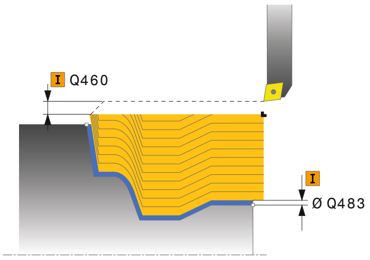
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	**Q485 Allowance for workpiece blank?** Contour-parallel oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q486 Type of cut lines (=0/1)? Define the type of cutting lines: **0**: Cuts with consistent chip cross section **1**: Equidistance cut distribution Input: **0**, **1**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Example

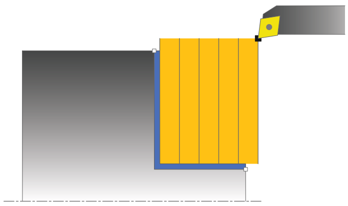
```
11 CYCL DEF 815 CONTOUR-PAR. TURNING ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q485=+5                ;ALLOWANCE ON BLANK ~
    Q486=+0                ;INTERSECTING LINES ~
    Q499=+0                ;REVERSE CONTOUR ~
    Q463=+3                ;MAX. CUTTING DEPTH ~
    Q478=0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.13 Cycle 821 SHOULDER, FACE

ISO programming
G821

Application

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



This cycle enables you to face turn right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1. The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. The control performs a paraxial infeed movement at rapid traverse.
3. The control finishes the contour of the finished part at the defined feed rate **Q505**.
4. The control retracts the tool at the defined feed rate to the set-up clearance.
5. The control returns the tool at rapid traverse to the cycle starting point.

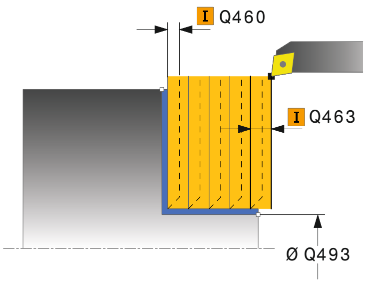
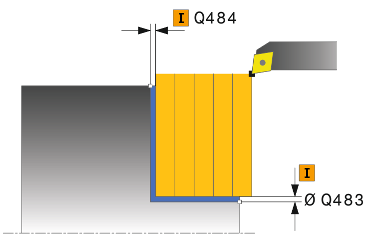
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

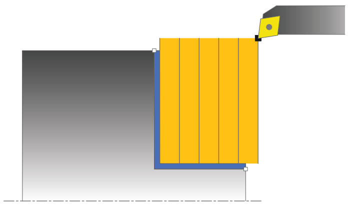
```
11 CYCL DEF 821 SHOULDER, FACE ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q493=+30             ;DIAMETER AT CONTOUR END ~
    Q494=-5              ;CONTOUR END IN Z ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q506=+0              ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.14 Cycle 822 SHOULDER, FACE. EXT.

ISO programming
G822

Application

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



This cycle enables you to face turn shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the Z coordinate and then in the X coordinate to set-up clearance and begins the cycle there.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1. The control performs a paraxial infeed movement at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

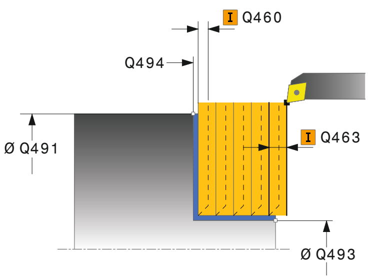
Notes

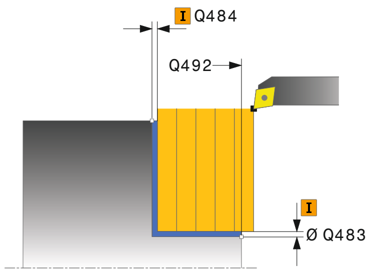
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

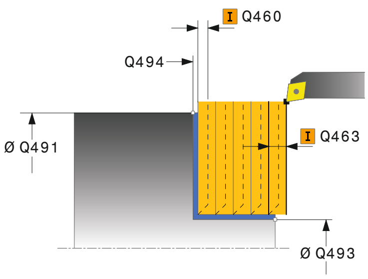
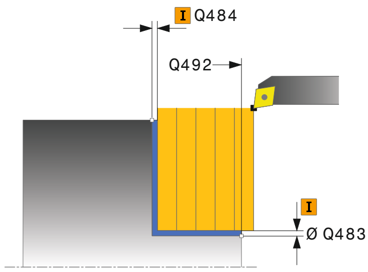
- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters





Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of the face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

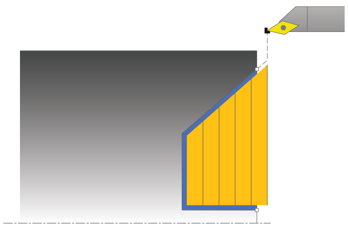
```
11 CYCL DEF 822 SHOULDER, FACE. EXT. ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q491=+75                 ;DIAMETER AT CONTOUR START ~
    Q492=+0                  ;CONTOUR START IN Z ~
    Q493=+30                 ;DIAMETER AT CONTOUR END ~
    Q494=-15                 ;CONTOUR END IN Z ~
    Q495=+0                  ;ANGLE OF FACE ~
    Q501=+1                  ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5                ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5                ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                  ;ANGLE OF CIRCUM. SURFACE ~
    Q503=+1                  ;TYPE OF END ELEMENT ~
    Q504=+0.5                ;SIZE OF END ELEMENT ~
    Q463=+3                  ;MAX. CUTTING DEPTH ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q506=+0                  ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.15 Cycle 823 TURN TRANSVERSE PLUNGE

ISO programming
G823

Application

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



This cycle enables you to run face turning of plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3 The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

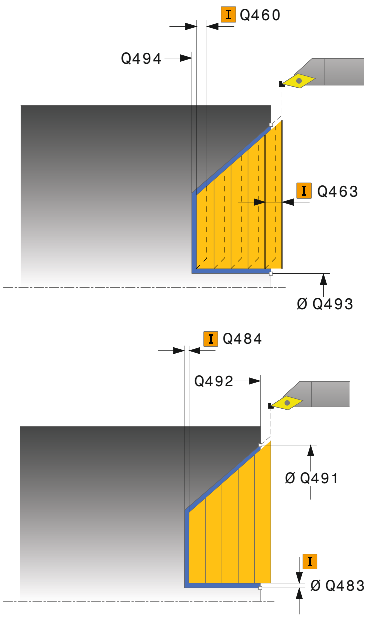
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters



Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point for the plunging path Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle of plunging flank. The reference angle is a line parallel to the rotary axis. Input: **0...89.9999**
	Q463 Maximum cutting depth? Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**

Help graphic	Parameter
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

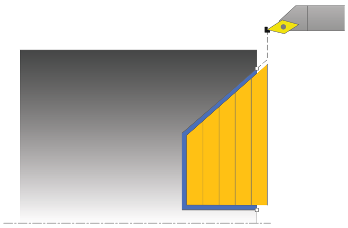
```
11 CYCL DEF 823 TURN TRANSVERSE PLUNGE ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q493=+20                ;DIAMETER AT CONTOUR END ~
    Q494=-5                 ;CONTOUR END IN Z ~
    Q495=+60                ;ANGLE OF SIDE ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.16 Cycle 824 TURN PLUNGE TRANSVERSE EXT.

ISO programming
G824

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run face turning of plunging elements (undercuts).
Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3. The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The infeed movement is performed at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

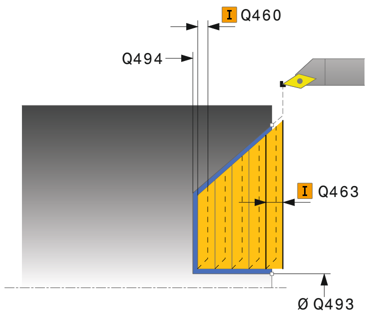
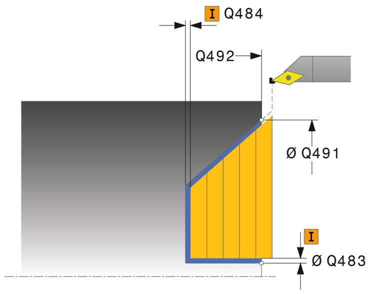
Notes

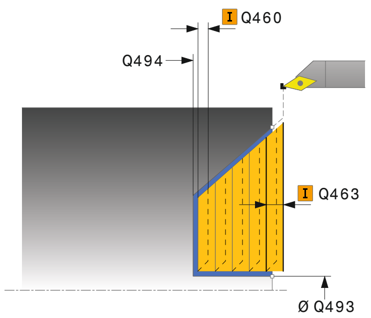
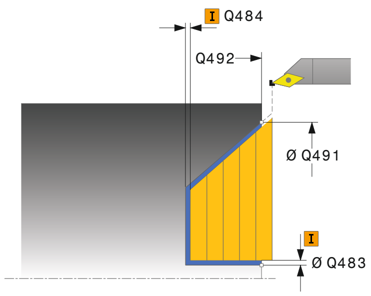
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the starting point for the plunging path (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point for the plunging path Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle of plunging flank. The reference angle is a line parallel to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

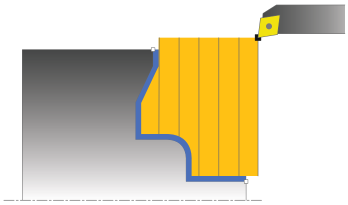
```
11 CYCL DEF 824 TURN PLUNGE TRANSVERSE EXT. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q493=+20                ;DIAMETER AT CONTOUR END ~
    Q494=-10                ;CONTOUR END IN Z ~
    Q495=+70                ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+0                 ;ANGLE OF FACE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.17 Cycle 820 TURN CONTOUR TRANSV.

ISO programming
G820

Application

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



This cycle enables you to run face turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction. The transverse cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

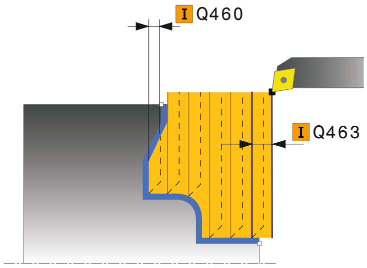
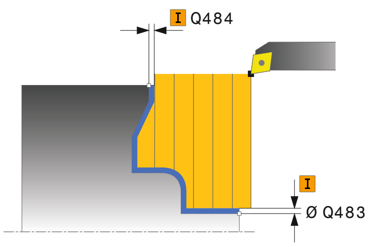
- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 760

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
 	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	Q463 Maximum cutting depth? Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
 	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q487 Allow plunging (0/1)? Permit the machining of plunging elements: **0**: Do not machine any plunging elements **1**: Machine plunging elements Input: **0**, **1**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

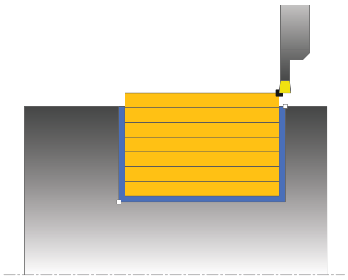
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 820 TURN CONTOUR TRANSV. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q499=+0                 ;REVERSE CONTOUR ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q487=+1                 ;PLUNGE ~
    Q488=+0                 ;PLUNGING FEED RATE ~
    Q479=+0                 ;CONTOUR MACHINING LIMIT ~
    Q480=+0                 ;DIAMETER LIMIT VALUE ~
    Q482=+0                 ;LIMIT VALUE Z ~
    Q506=+0                 ;CONTOUR SMOOTHING
14 L  X+75  Y+0  Z+2 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+75  Z-20
19 L  X+50
20 RND R2
21 L  X+20  Z-25
22 RND R2
23 L  Z+0
24 LBL 0
```

15.4.18 Cycle 841 SIMPLE REC. TURNG., RADIAL DIR.

ISO programming
G841

Application

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



This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

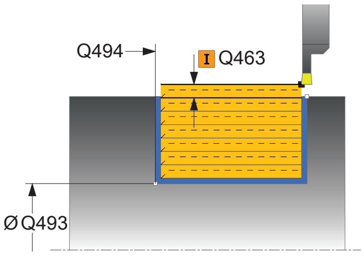
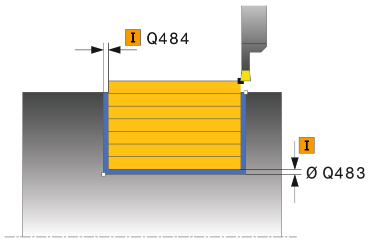
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**

Help graphic	Parameter
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

Example

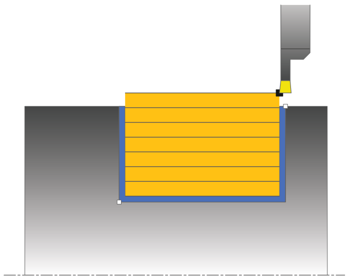
```
11 CYCL DEF 841 SIMPLE REC. TURNG., RADIAL DIR.. ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q493=+50                 ;DIAMETER AT CONTOUR END ~
    Q494=-50                 ;CONTOUR END IN Z ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q463=+2                  ;MAX. CUTTING DEPTH ~
    Q507=+0                  ;MACHINING DIRECTION ~
    Q508=+0                  ;OFFSET WIDTH ~
    Q509=+0                  ;DEPTH COMPENSATION ~
    Q488=+0                  ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.19 Cycle 842 ENH.REC.TURNNG, RAD.

ISO programming
G842

Application

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



This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 Diameter at contour start**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1 From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the position of the tool at the cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 DIAMETER AT CONTOUR START**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

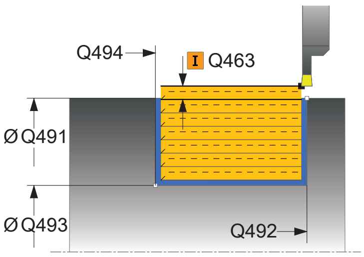
Notes

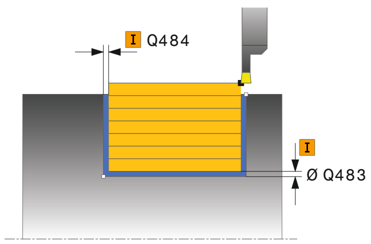
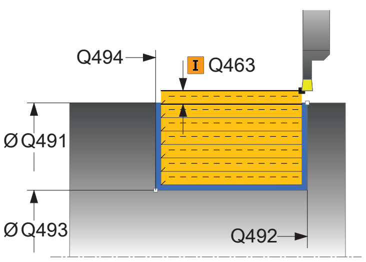
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and the normal line to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge at the contour end point and the normal line to the rotary axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**

Help graphic	Parameter
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

Example

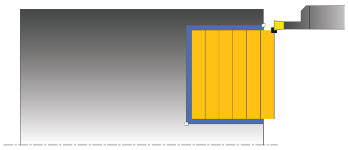
```
11 CYCL DEF 842 EXPND. RECESS, RADL. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=-20                ;CONTOUR START IN Z ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-50                ;CONTOUR END IN Z ~
    Q495=+5                 ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                 ;ANGLE OF SECOND SIDE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q463=+2                 ;MAX. CUTTING DEPTH ~
    Q507=+0                 ;MACHINING DIRECTION ~
    Q508=+0                 ;OFFSET WIDTH ~
    Q509=+0                 ;DEPTH COMPENSATION ~
    Q488=+0                 ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.20 Cycle 851 SIMPLE REC TURNG, AX

ISO programming
G851

Application

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



This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1 From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

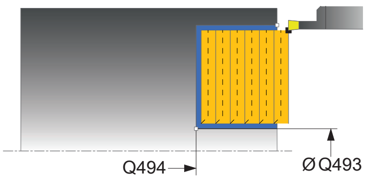
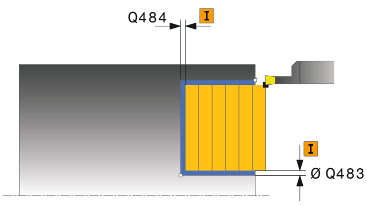
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**

Help graphic	Parameter
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

Example

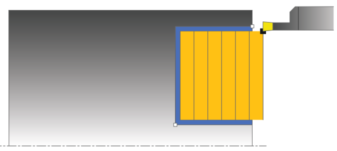
```
11 CYCL DEF 851 SIMPLE REC TURNG, AX ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-10                ;CONTOUR END IN Z ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q463=+2                 ;MAX. CUTTING DEPTH ~
    Q507=+0                 ;MACHINING DIRECTION ~
    Q508=+0                 ;OFFSET WIDTH ~
    Q509=+0                 ;DEPTH COMPENSATION ~
    Q488=+0                 ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.21 Cycle 852 ENH.REC.TURNING, AX.

ISO programming
G852

Application

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



This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed. The machining process thus requires a minimum of retraction and infeed movements. Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

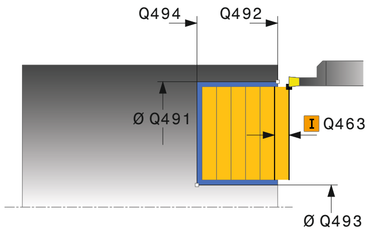
Notes

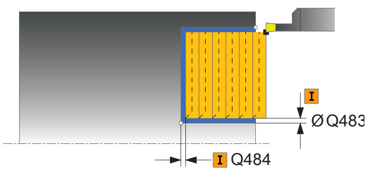
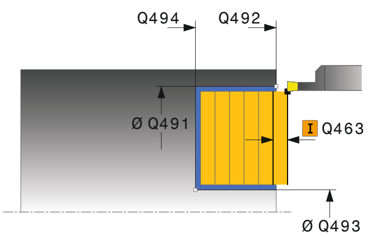
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and a line parallel to the turning axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge of the contour end point and a line parallel to the turning axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**

Help graphic	Parameter
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

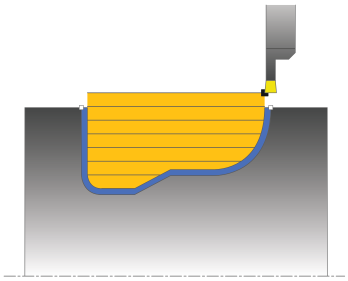
Example

```
11 CYCL DEF 852 ENH.REC.TURNING, AX. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=-20                ;CONTOUR START IN Z ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-50                ;CONTOUR END IN Z ~
    Q495=+5                 ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                 ;ANGLE OF SECOND SIDE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q463=+2                 ;MAX. CUTTING DEPTH ~
    Q507=+0                 ;MACHINING DIRECTION ~
    Q508=+0                 ;OFFSET WIDTH ~
    Q509=+0                 ;DEPTH COMPENSATION ~
    Q488=+0                 ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.22 Cycle 840 RECESS TURNG, RADIAL

ISO programming
G840

Application



This cycle enables you to recess slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the X coordinate of the starting point is less than the contour starting point, the control positions the tool in the X coordinate to the contour starting point and begins the cycle there.

1. The control positions the tool at rapid traverse in the Z coordinate (first recessing position).
2. The control performs a recessing traverse until the first plunging depth is reached.
3. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
4. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6. The tool recesses to the next plunging depth.
7. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side walls of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate.
4. The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

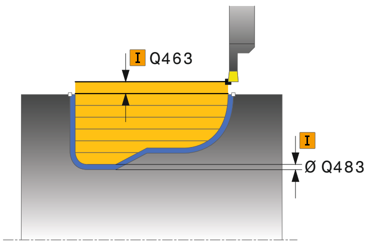
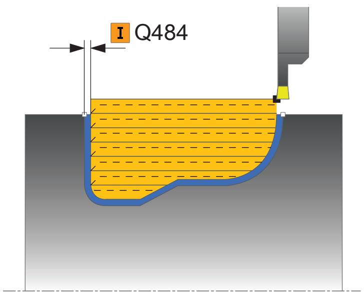
▶ Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	**Q484 Oversize in Z?** Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**

Help graphic	Parameter
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q499 Reverse contour (0=no/1=yes)? Machining direction: **0**: Machining in the direction of contour **1**: Machining in the direction opposite to the contour direction Input: **0**, **1**

Example

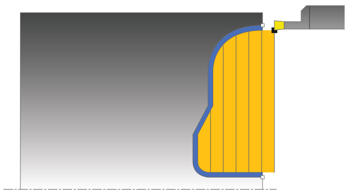
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 840 RECESS TURNG, RADIAL ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q488=+0              ;PLUNGING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q479=+0              ;CONTOUR MACHINING LIMIT ~
    Q480=+0              ;DIAMETER LIMIT VALUE ~
    Q482=+0              ;LIMIT VALUE Z ~
    Q463=+2              ;MAX. CUTTING DEPTH ~
    Q507=+0              ;MACHINING DIRECTION ~
    Q508=+0              ;OFFSET WIDTH ~
    Q509=+0              ;DEPTH COMPENSATION ~
    Q499=+0              ;REVERSE CONTOUR
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z-10
19 L  X+40  Z-15
20 RND R3
21 CR  X+40  Z-35 R+30 DR+
22 RND R3
23 L  X+60  Z-40
24 LBL 0
```

15.4.23 Cycle 850 RECESS TURNG, AXIAL

ISO programming
G850

Application

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



This cycle enables you to machine slots of any shape in transverse direction by recess turning. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1. The control positions the tool at rapid traverse in the X coordinate (first recessing position).
2. The control performs a recessing traverse until the first plunging depth is reached.
3. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
4. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6. The tool recesses to the next plunging depth.
7. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side walls of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate.
4. The control returns the tool at rapid traverse to the cycle starting point.

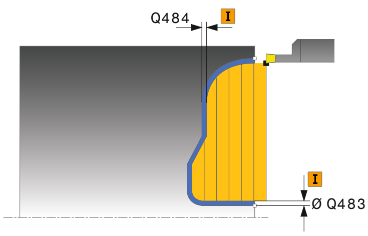
Notes

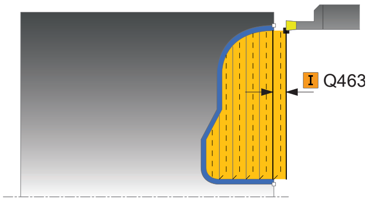
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**

Help graphic	Parameter
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q499 Reverse contour (0=no/1=yes)? Machining direction: **0**: Machining in the direction of contour **1**: Machining in the direction opposite to the contour direction Input: **0**, **1**

Example

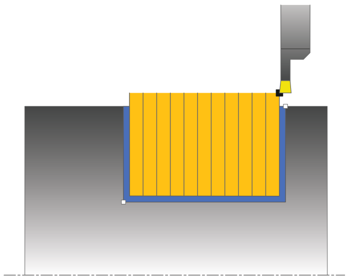
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 850 RECESS TURNG, AXIAL ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q488=0                  ;PLUNGING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q479=+0                 ;CONTOUR MACHINING LIMIT ~
    Q480=+0                 ;DIAMETER LIMIT VALUE ~
    Q482=+0                 ;LIMIT VALUE Z ~
    Q463=+2                 ;MAX. CUTTING DEPTH ~
    Q507=+0                 ;MACHINING DIRECTION ~
    Q508=+0                 ;OFFSET WIDTH ~
    Q509=+0                 ;DEPTH COMPENSATION ~
    Q499=+0                 ;REVERSE CONTOUR
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Y-15
22 L  Z+0
23 LBL 0
```

15.4.24 Cycle 861 SIMPLE RECESS, RADL.

ISO programming
G861

Application

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



This cycle enables you to radially cut in right-angled slots.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle sequence

The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

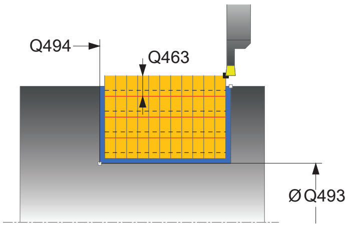
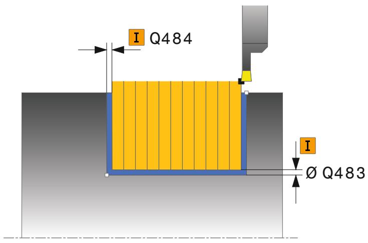
Notes

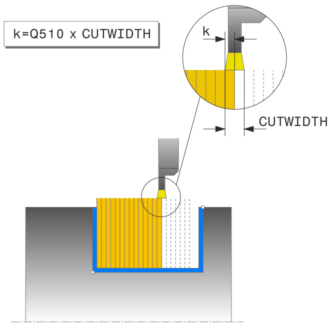
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
 	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
 	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**

Help graphic	Parameter
	**Q510 Overlap factor for recess width?** Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

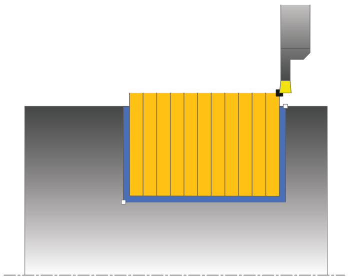
```
11 CYCL DEF 861 SIMPLE RECESS, RADL. ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q493=+50                 ;DIAMETER AT CONTOUR END ~
    Q494=-50                 ;CONTOUR END IN Z ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q463=+0                  ;LIMIT TO DEPTH ~
    Q510=+0.8                ;RECESSING OVERLAP ~
    Q511=+100                ;FEED RATE FACTOR ~
    Q462=0                   ;RETRACTION MODE ~
    Q211=3                   ;DWELL TIME IN REVS ~
    Q562=+0                  ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.25 Cycle 862 EXPND. RECESS, RADL.

ISO programming
G862

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to radially cut in slots. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle sequence

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes half the slot width at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control finishes half the slot width at the defined feed rate.
8. The control returns the tool at rapid traverse to the cycle starting point.

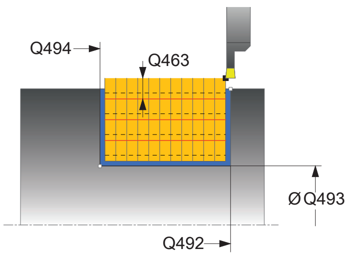
Notes

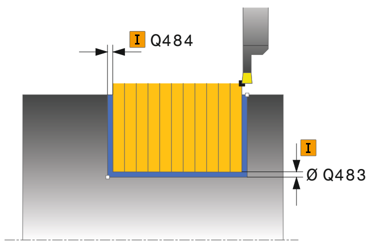
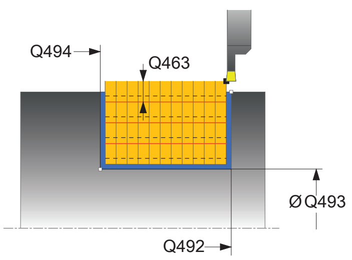
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

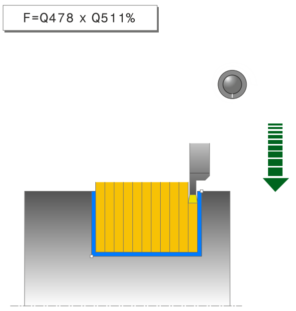
Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	**Q492 Contour start in Z?** Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and the normal line to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge at the contour end point and the normal line to the rotary axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	**Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

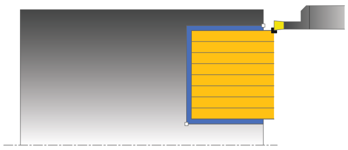
```
11 CYCL DEF 862 EXPND. RECESS, RADL. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=-20                ;CONTOUR START IN Z ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-50                ;CONTOUR END IN Z ~
    Q495=+5                 ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                 ;ANGLE OF SECOND SIDE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q463=+0                 ;LIMIT TO DEPTH ~
    Q510=0.8                ;RECESSING OVERLAP ~
    Q511=+100               ;FEED RATE FACTOR ~
    Q462=+0                 ;RETRACTION MODE ~
    Q211=3                  ;DWELL TIME IN REVS ~
    Q562=+0                 ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.26 Cycle 871 SIMPLE RECESS, AXIAL

ISO programming
G871

Application

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



This cycle enables you to perform axial recessing of right-angled slots (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

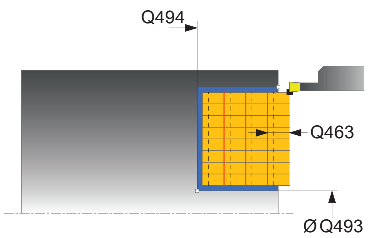
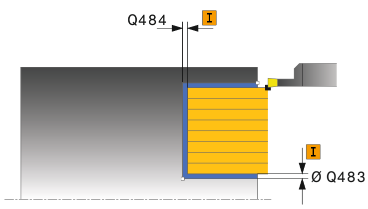
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

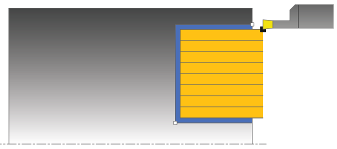
```
11 CYCL DEF 871 SIMPLE RECESS, AXIAL ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q493=+50                 ;DIAMETER AT CONTOUR END ~
    Q494=-10                 ;CONTOUR END IN Z ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q463=+0                  ;LIMIT TO DEPTH ~
    Q510=+0,8                ;RECESSING OVERLAP ~
    Q511=+100                ;FEED RATE FACTOR ~
    Q462=0                   ;RETRACTION MODE ~
    Q211=3                   ;DWELL TIME IN REVS ~
    Q562=+0                  ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.27 Cycle 872 EXPND. RECESS, AXIAL

ISO programming
G872

Application

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



This cycle enables you to perform axial recessing of slots (face recessing). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control retracts the tool at rapid traverse.
4 The control positions the tool at rapid traverse to the second slot side.
5 The control finishes the side wall of the slot at the defined feed rate **Q505**.
6 The control finishes one half of the slot at the defined feed rate.
7 The control positions the tool at rapid traverse to the first side.
8 The control finishes the other half of the slot at the defined feed rate.
9 The control returns the tool at rapid traverse to the cycle starting point.

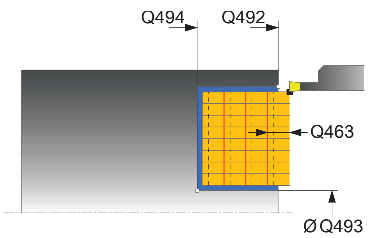
Notes

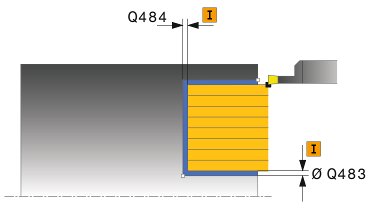
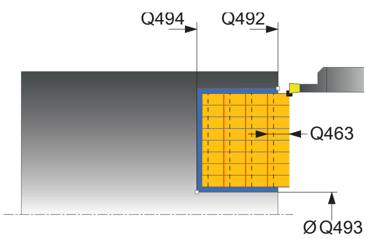
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	**Q492 Contour start in Z?** Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and a line parallel to the turning axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge of the contour end point and a line parallel to the turning axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

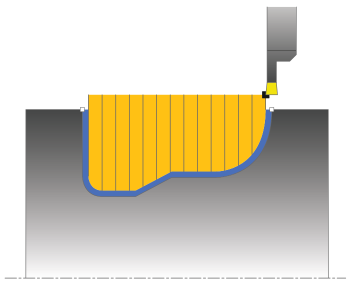
```
11 CYCL DEF 872 EXPND. RECESS, AXIAL ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q491=+75               ;DIAMETER AT CONTOUR START ~
    Q492=-20               ;CONTOUR START IN Z ~
    Q493=+50               ;DIAMETER AT CONTOUR END ~
    Q494=-50               ;CONTOUR END IN Z ~
    Q495=+5                ;ANGLE OF SIDE ~
    Q501=+1                ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5              ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5              ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                ;ANGLE OF SECOND SIDE ~
    Q503=+1                ;TYPE OF END ELEMENT ~
    Q504=+0.5              ;SIZE OF END ELEMENT ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q463=+0                ;LIMIT TO DEPTH ~
    Q510=+0.08             ;RECESSING OVERLAP ~
    Q511=+100              ;FEED RATE FACTOR ~
    Q462=+0                ;RETRACTION MODE ~
    Q211=+3                ;DWELL TIME IN REVS ~
    Q562=+0                ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.28 Cycle 860 CONT. RECESS, RADIAL

ISO programming
G860

Application

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



This cycle enables you to radially cut in slots of any form.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle sequence

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

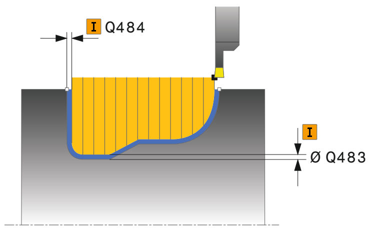
▶ Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

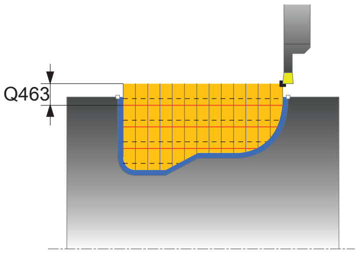
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**

Help graphic	Parameter
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

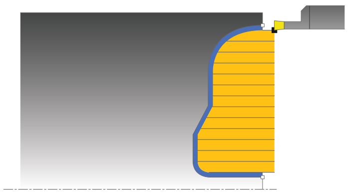
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 860 CONT. RECESS, RADIAL ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q479=+0                 ;CONTOUR MACHINING LIMIT ~
    Q480=+0                 ;DIAMETER LIMIT VALUE ~
    Q482=+0                 ;LIMIT VALUE Z ~
    Q463=+0                 ;LIMIT TO DEPTH ~
    Q510=0.08               ;RECESSING OVERLAP ~
    Q511=+100               ;FEED RATE FACTOR ~
    Q462=+0                 ;RETRACTION MODE ~
    Q211=3                  ;DWELL TIME IN REVS ~
    Q562=+0                 ;MULTIPLE PLUNGING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z-20
19 L  X+45
20 RND R2
21 L  X+40  Y-25
22 L  Z+0
23 LBL 0
```

15.4.29 Cycle 870 CONT. RECESS, AXIAL

ISO programming
G870

Application

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



This cycle enables you to perform axial recessing of slots of any form (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes one half of the slot at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control finishes the other half of the slot at the defined feed rate.
8. The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

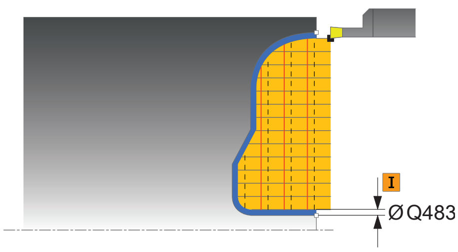
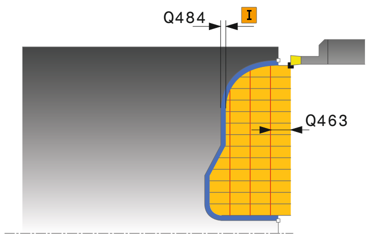
▶ Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	**Q484 Oversize in Z?** Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**

Help graphic	Parameter
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

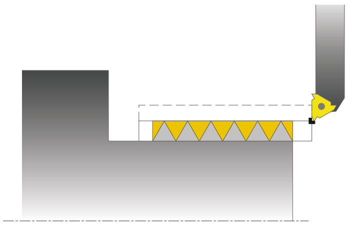
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 870 CONT. RECESS, AXIAL ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q479=+0                  ;CONTOUR MACHINING LIMIT ~
    Q480=+0                  ;DIAMETER LIMIT VALUE ~
    Q482=+0                  ;LIMIT VALUE Z ~
    Q463=+0                  ;LIMIT TO DEPTH ~
    Q510=+0.8                ;RECESSING OVERLAP ~
    Q511=+100                ;FEED RATE FACTOR ~
    Q462=+0                  ;RETRACTION MODE ~
    Q211=+3                  ;DWELL TIME IN REVS ~
    Q562=+0                  ;MULTIPLE PLUNGING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Y-15
22 L  Z+0
23 LBL 0
```

15.4.30 Cycle 831 THREAD LONGITUDINAL

ISO programming
G831

Application

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



This cycle enables you to run longitudinal turning of threads.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth, the cycle uses thread depth in accordance with the ISO1502 standard.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a paraxial longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

	While the control cuts a thread, the feed-rate override knob is disabled. The feed-rate override knob is still active to a limited extent.

Notes

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g., to machine internal and external threads with only one tool. If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position. There is a danger of collision!

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Caution: Danger to the tool and workpiece!

If you program an angle of infeed **Q467** wider than the side angle of the thread, this may destroy the thread flanks. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

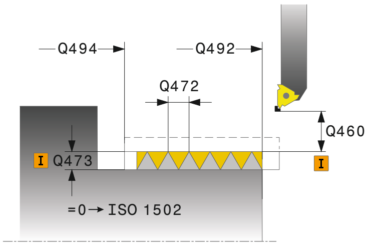
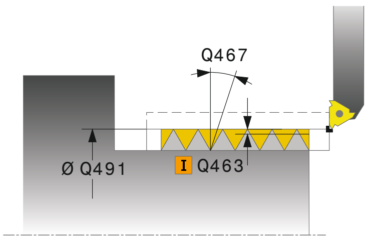
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The number of threads for thread cutting is limited to 500.
- In Cycle **832 THREAD EXTENDED**, parameters are available for approach and overrun.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The control uses the set-up clearance **Q460** as approach length. The approach path must be long enough for the feed axes to be accelerated to the required velocity.
- The control uses the thread pitch as idle travel path. The idle travel distance must be long enough to decelerate the feed axes.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.

Cycle parameters

Help graphic	Parameter
	Q471 Thread position (0=ext./1=int.)? Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	**Q460 Setup clearance?** Set-up clearance in radial and axial direction. In axial direction, the set-up clearance is used for acceleration (approach path) until the synchronized feed rate is reached. Input: **0...999.999**
	Q491 Thread diameter? Define the nominal diameter of the thread. Input: **0.001...99999.999**,
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	Q492 Contour start in Z? Z coordinate of the starting point Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the end point, including the thread runout **Q474** Input: **-99999.999...+99999.999**
	Q474 Length of thread runout? Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q463 Maximum cutting depth? Maximum plunging depth in radial direction relative to the radius. Input: **0,001...999.999**
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is the line perpendicular to the rotary axis. Input: **0...60**

Help graphic	Parameter
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

Example

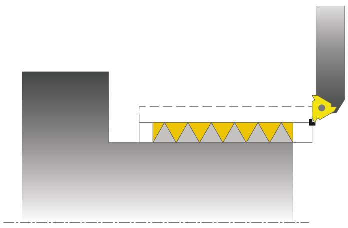
```
11 CYCL DEF 831 THREAD LONGITUDINAL ~
    Q471=+0                 ;THREAD POSITION ~
    Q460=+5                 ;SAFETY CLEARANCE ~
    Q491=+75                ;THREAD DIAMETER ~
    Q472=+2                 ;THREAD PITCH ~
    Q473=+0                 ;DEPTH OF THREAD ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q494=-15                ;CONTOUR END IN Z ~
    Q474=+0                 ;THREAD RUN-OUT ~
    Q463=+0.5               ;MAX. CUTTING DEPTH ~
    Q467=+30                ;ANGLE OF INFEED ~
    Q468=+0                 ;TYPE OF INFEED ~
    Q470=+0                 ;STARTING ANGLE ~
    Q475=+30                ;NUMBER OF STARTS ~
    Q476=+30                ;NUMBER OF AIR CUTS
12 L  X+80  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.31 Cycle 832 THREAD EXTENDED

ISO programming
G832

Application

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



This cycle enables you to run both face turning and longitudinal turning of threads or tapered threads. Expanded scope of function:

- Selection of a longitudinal thread or transversal thread
- The parameters for dimension type of taper, taper angle, and contour starting point X enable the definition of various tapered threads
- The parameters for the approach length and the idle travel distance define a path in which feed axes can be accelerated and decelerated

You can process single threads or multi-threads with the cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

While the control cuts a thread, the feed-rate override knob is disabled. The feed-rate override knob is still active to a limited extent.

Notes

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g., to machine internal and external threads with only one tool. If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position. There is a danger of collision!

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Caution: Danger to the tool and workpiece!

If you program an angle of infeed **Q467** wider than the side angle of the thread, this may destroy the thread flanks. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

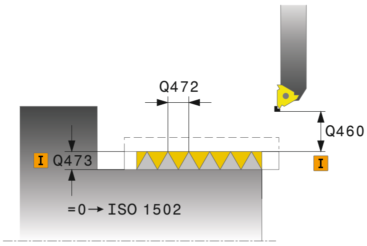
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.
- The overrun path (**Q466**) must be long enough to decelerate the feed axes.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.

Cycle parameters

Help graphic	Parameter
	Q471 Thread position (0=ext./1=int.)? Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	Q461 Thread orientation (0/1)? Define the direction of the thread pitch: **0**: L (parallel to the turning axis) **1**: Perpendicular (perpendicular to the turning axis) Input: **0**, **1**
 	**Q460 Set-up clearance?** Set-up clearance perpendicular to the thread pitch Input: **0...999.999**
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	Q464 Dimens. type taper (0-4)? Type of dimensioning of the taper contour: **0**: Via start and end point **1**: Via end point, start X and angle of taper **2**: Via end point, start Z and angle of taper **3**: Via start point, end X and angle of taper **4**: Via start point, end Z and angle of taper Input: **0**, **1**, **2**, **3**, **4**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the end point Input: **-99999.999...+99999.999**

Help graphic	Parameter
	Q469 Taper angle (diameter)? Taper angle of the contour Input: **-180...+180**
	Q474 Length of thread runout? Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q465 Starting path? Length of the path in the direction of the pitch at which the feed axes are accelerated to the required speed. The approach path is outside of the defined thread contour. This value has an incremental effect. Input: **0.1...99.9**
	Q466 Overrun path? Input: **0.1...99.9**
	Q463 Maximum cutting depth? Maximum infeed perpendicular to the thread pitch Input: **0,001...999.999**
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is formed by the parallel line to the thread pitch. Input: **0...60**
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

Example

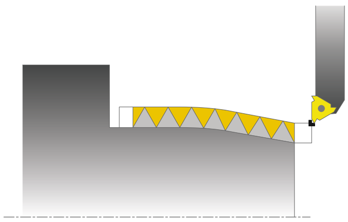
```
11 CYCL DEF 832 THREAD EXTENDED ~
    Q471=+0          ;THREAD POSITION ~
    Q461=+0          ;THREAD ORIENTATION ~
    Q460=+2          ;SAFETY CLEARANCE ~
    Q472=+2          ;THREAD PITCH ~
    Q473=+0          ;DEPTH OF THREAD ~
    Q464=+0          ;DIMENSION TYPE TAPER ~
    Q491=+100        ;DIAMETER AT CONTOUR START ~
    Q492=+0          ;CONTOUR START IN Z ~
    Q493=+110        ;DIAMETER AT CONTOUR END ~
    Q494=-35         ;CONTOUR END IN Z ~
    Q469=+0          ;TAPER ANGLE ~
    Q474=+0          ;THREAD RUN-OUT ~
    Q465=+4          ;STARTING PATH ~
    Q466=+4          ;OVERRUN PATH ~
    Q463=+0.5        ;MAX. CUTTING DEPTH ~
    Q467=+30         ;ANGLE OF INFEED ~
    Q468=+0          ;TYPE OF INFEED ~
    Q470=+0          ;STARTING ANGLE ~
    Q475=+30         ;NUMBER OF STARTS ~
    Q476=+30         ;NUMBER OF AIR CUTS
12 L  X+80  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

15.4.32 Cycle 830 THREAD CONTOUR-PARALLEL

ISO programming
G830

Application

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



This cycle enables you to run both face turning and longitudinal turning of threads with any shape.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control runs a thread cut parallel to the defined thread contour. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

While the control cuts a thread, the feed-rate override knob is disabled. The feed-rate override knob is still active to a limited extent.

Notes

NOTICE

Danger of collision!

Cycle **830** executes the overrun **Q466** following the programmed contour. There is a danger of collision!

- Clamp the workpiece in such a way that there is no danger of collision if the control extends the contour by **Q466**, **Q467**.

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g., to machine internal and external threads with only one tool. If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position. There is a danger of collision!

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Caution: Danger to the tool and workpiece!

If you program an angle of infeed **Q467** wider than the side angle of the thread, this may destroy the thread flanks. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

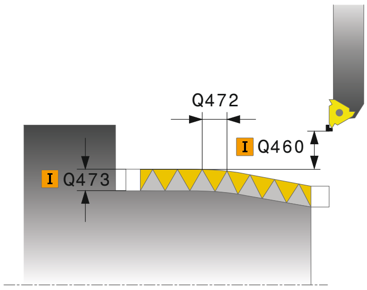
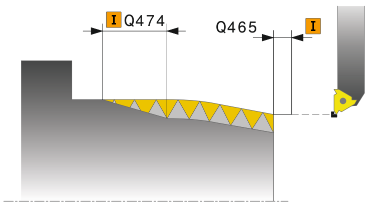
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- Both the approach and overrun take place outside the defined contour.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.
- The overrun path (**Q466**) must be long enough to decelerate the feed axes.
- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

<table>
<tr><th>Help graphic</th><th>Parameter</th></tr>
<tr><td></td><td>Q471 Thread position (0=ext./1=int.)?
Define the position of the thread:
0: External thread
1: Internal thread
Input: 0, 1</td></tr>
<tr><td></td><td>Q461 Thread orientation (0/1)?
Define the direction of the thread pitch:
0: L (parallel to the turning axis)
1: Perpendicular (perpendicular to the turning axis)
Input: 0, 1</td></tr>
<tr><td></td><td>Q460 Set-up clearance?
Set-up clearance perpendicular to the thread pitch
Input: 0...999.999</td></tr>
<tr><td></td><td>Q472 Thread pitch?
Pitch of the thread
Input: 0...99999.999</td></tr>
<tr><td></td><td>Q473 Thread depth (radius)?
Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect.
Input: 0...999.999</td></tr>
<tr><td></td><td>Q474 Length of thread runout?
Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460. This value has an incremental effect.
Input: 0...999.999</td></tr>
<tr><td></td><td>Q465 Starting path?
Length of the path in the direction of the pitch at which the feed axes are accelerated to the required speed. The approach path is outside of the defined thread contour. This value has an incremental effect.
Input: 0.1...99.9</td></tr>
<tr><td></td><td>Q466 Overrun path?
Input: 0.1...99.9</td></tr>
<tr><td></td><td>Q463 Maximum cutting depth?
Maximum infeed perpendicular to the thread pitch
Input: 0,001...999.999</td></tr>
</table>

Help graphic	Parameter
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is formed by the parallel line to the thread pitch. Input: **0...60**
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

Example

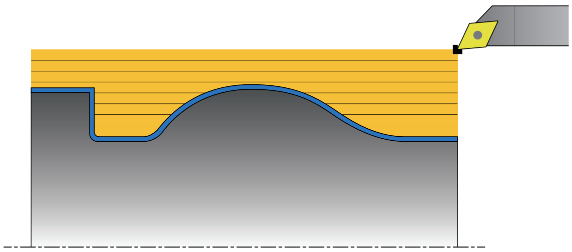
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 830 THREAD CONTOUR-PARALLEL ~
    Q471=+0                  ;THREAD POSITION ~
    Q461=+0                  ;THREAD ORIENTATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q472=+2                  ;THREAD PITCH ~
    Q473=+0                  ;DEPTH OF THREAD ~
    Q474=+0                  ;THREAD RUN-OUT ~
    Q465=+4                  ;STARTING PATH ~
    Q466=+4                  ;OVERRUN PATH ~
    Q463=+0.5                ;MAX. CUTTING DEPTH ~
    Q467=+30                 ;ANGLE OF INFEED ~
    Q468=+0                  ;TYPE OF INFEED ~
    Q470=+0                  ;STARTING ANGLE ~
    Q475=+30                 ;NUMBER OF STARTS ~
    Q476=+30                 ;NUMBER OF AIR CUTS
14 L  X+80  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  X+70  Z-30
20 RND R60
21 L  Z-45
22 LBL 0
```

15.4.33 Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option158)

ISO programming
G882

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



In Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**, the defined contour area is roughed simultaneously in several steps using a movement that includes at least 3 axes (two linear axes and one rotary axis). This allows machining of complex contours with a single tool. During machining, the cycle continuously adjusts the tool angle of inclination based on the following criteria:

- Avoiding collisions between the workpiece, the tool, and the tool carrier
- The tooth does not suffer single-spot wear
- Undercuts are possible

Execution with a FreeTurn tool

You can execute this cycle with FreeTurn tools. This method allows you to perform the most common turning operations with just one tool. Machining times can be reduced through the flexible tool because fewer tool changes occur.

Requirements:

- This function must be enabled by your machine manufacturer.
- You must properly define the tool.
  Further information: "Turning operation with FreeTurn tools", Page 244

ℹ	The NC program remains unchanged except for the calling of the FreeTurn cutting edges, see "Example: Turning with a FreeTurn tool", Page 906

Roughing cycle sequence

1 The cycle positions the tool at the cycle start position (tool position when the cycle is called), taking the first tool angle of inclination into account. Then, the tool moves to set-up clearance. If the angle of inclination cannot be achieved at the cycle start position, the control first moves the tool to set-up clearance and from there tilts it using the first tool angle of inclination.
2 The tool moves to the plunging depth **Q519**. The profile infeed may be exceeded for a short time up to the value of **Q463 MAX. CUTTING DEPTH**, e.g. in the case of a corner.
3 The contour is roughed simultaneously using the roughing feed-rate in **Q478**. If you define the plunging feed rate **Q488** in the cycle, it will be effective for the plunging elements. Machining depends on the following input parameters:
   - **Q590**: **MACHINING MODE**
   - **Q591**: **MACHINING SEQUENCE**
   - **Q389**: **UNI.- BIDIRECTIONAL**
4 After each infeed, the control lifts the tool in rapid traverse by the set-up clearance value.
5 The control repeats steps 2 to 4 until the contour has been machined completely.
6 The control retracts the tool at the machining feed rate by the set-up clearance value and then moves it with rapid traverse to the starting position (first in the X axis and then in the Z axis direction)

Notes

NOTICE

Danger of collision!

The control does not perform collision monitoring (DCM). Danger of collision during machining!

- ▶ Run a simulation to verify the sequence and the contour
- ▶ Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

The cycle uses the position of the tool at cycle call as the cycle starting position. Incorrect pre-positioning can cause contour damage. There is a danger of collision!

- ▶ Move the tool to a safe position in the X and Z axes.

NOTICE

Danger of collision!

If the contour ends too closely at the fixture, a collision between tool and fixture might occur during machining.

- ▶ When clamping, take both the tool angle of inclination and the departure movement into account

NOTICE

Danger of collision!

Collision monitoring only considers the two-dimensional X-Z working plane. The cycle does not check for collisions with an area in the Y coordinate of the cutting edge, tool holder, or tilting body.

- Verify the NC program in **Program Run** in **Single Block**
- Limit the machining area

NOTICE

Danger of collision!

Depending on the geometry of the cutting edge, residual material may be left over. Danger of collision during subsequent machining operations!

- Run a simulation to verify the sequence and the contour

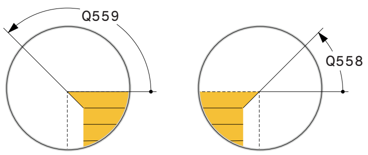
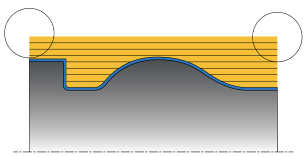
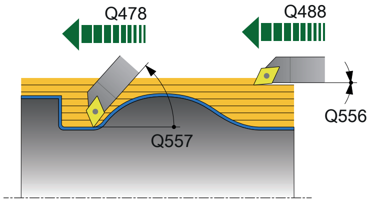
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- If you programmed **M136** before the cycle call, the control interprets the feed rate in millimeters per revolution.
- Software limit switches limit the possible inclination angle **Q556** and **Q557**. If, in **Editor** in the **Simulation** the switch for the software end switches is deactivated, then the simulation may deviate from the later machining operation.
- If it is not possible to machine a particular contour area using this cycle, the control tries to divide the contour area into subareas that can be reached so as to machine them individually.

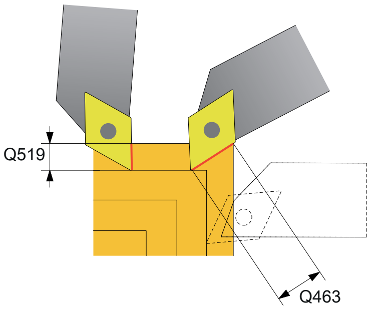
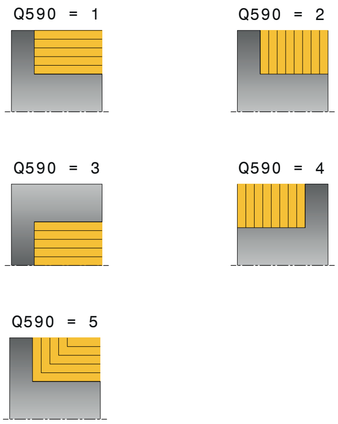
Notes on programming

- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- Prior to the cycle call, you must program **FUNCTION TCPM**. In **FUNCTION TCMP**, HEIDENHAIN recommends programming the tool reference point **REFPNT TIP-CENTER**.
- The cycle requires a radius compensation (**RL**/**RR**) in its contour description.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- For determining the inclination angle, the cycle requires the definition of a tool holder. For this purpose, assign a tool holder to the tool in the **KINEMATIC** column of the tool table.

  Further information: "Tool management ", Page 297
- Define a value in **Q463 MAX. CUTTING DEPTH** relative to the cutting edge because, depending on the tool inclination, the infeed from **Q519** may be temporarily exceeded. Use this parameter to limit the extent to which the infeed may be exceeded.

Cycle parameters

Help graphic	Parameter
	Q460 Set-up clearance? Retraction before and after a cut. And distance for the pre-positioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
  	**Q558 Extensn. angle at contour start?** Angle in the WPL-CS, by which the cycle extends the contour up to the workpiece blank at the programmed starting point. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	Q559 Extension angle at contour end? Angle in WPL CS by which the cycle extends the contour at the programmed end point up to the workpiece blank. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	Q478 Roughing feed rate? Feed rate during roughing in millimeters per minute Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging Feed rate in millimeters per minute for plunging. This input value is optional. If you do not program the feed rate for plunging, the roughing feed rate **Q478** will apply. Input: **0...99999.999** or **FAUTO**
	Q556 Minimum angle of inclination? Smallest possible permitted angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q557 Maximum angle of inclination? Largest possible angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q567 Finishing allowance of contour? Contour-parallel oversize that will remain after roughing. This value has an incremental effect. Input: **-9...99.999**

Help graphic	Parameter
	**Q519 Infeed on contour?** Axial, radial and contour-parallel infeed (per cut). Enter a value greater than 0. This value has an incremental effect. Input: **0,001...99.999**
	Q463 Maximum cutting depth? Limit of the maximum infeed relative to the cutting edge. Depending on the tool angle of inclination, the control may temporarily exceed the **Q519 INFEED**, e.g., when machining a corner. Use this optional parameter to limit the extent by which the infeed may be exceeded. If you define the value 0, the maximum infeed is two thirds of the length of the cutting edge. Input: **0...99.999**
	**Q590 Machining mode (0/1/2/3/4/5)?** Defining the direction of machining: **0**: Automatic; the control automatically combines transverse and longitudinal machining. **1**: Longitudinal turning (outside) **2**: Face turning (front face) **3**: Longitudinal turning (inside) **4**: Face turning (chuck) **5**: Contour-parallel Input: **0**, **1**, **2**, **3**, **4**, **5**
	Q591 Machining sequence (0/1)? Define the machining sequence after which the control executes the contour: **0**: Machining occurs in segments. The sequence is selected in such a way that the center of gravity of the workpiece is shifted towards the chuck as soon as possible. **1**: The workpiece is machined paraxially. The sequence is selected in such a way that the moment of inertia of the workpiece decreases as soon as possible. Input: **0**, **1**
	Q389 Machining strategy (0/1)? Definite the cutting direction: **0**: Unidirectional; every cut is made in the direction of the contour. The direction of the contour depends on **Q499** **1**: Bidirectional; cuts are made against the direction of the contour. The cycle determines the best direction for each following step. Input: **0**, **1**

Example

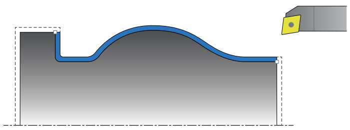
```
11 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q499=+0                 ;REVERSE CONTOUR ~
    Q558=+0                 ;EXT:ANGLE CONT.START ~
    Q559=+90                ;CONTOUR END EXT ANGL ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q488=+0.3               ;PLUNGING FEED RATE ~
    Q556=+0                 ;MIN. INCLINAT. ANGLE ~
    Q557=+90                ;MAX. INCLINAT. ANGLE ~
    Q567=+0.4               ;FINISH. ALLOW. CONT. ~
    Q519=+2                 ;INFEED ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q590=+0                 ;MACHINING MODE ~
    Q591=+0                 ;MACHINING SEQUENCE ~
    Q389=+1                 ;UNI.- BIDIRECTIONAL
12 L  X+58  Y+0 FMAX M303
13 L  Z+50 FMAX
14 CYCL CALL
```

15.4.34 Cycle 883 TURNING SIMULTANEOUS FINISHING (option 158)

ISO programming
G883

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer. The cycle is machine-dependent.



You can use this cycle to machine complex contours that are only accessible with different inclinations. When machining with this cycle, the inclination between tool and workpiece changes. This results in machining operations with at least 3 axes (two linear axes and one rotary axis).

The cycle monitors the workpiece contour with respect to the tool and the tool carrier. The cycle avoids unnecessary tilting movements in order to machine optimum surfaces.

If you want to force tilting movements, you can define inclination angles at the beginning and at the end of the contour. Even if simple contours have to be machined, you can use a large area of the indexable insert to achieve longer tool life.

Execution with a FreeTurn tool

You can execute this cycle with FreeTurn tools. This method allows you to perform the most common turning operations with just one tool. Machining times can be reduced through the flexible tool because fewer tool changes occur.

Requirements:

- This function must be enabled by your machine manufacturer.
- You must properly define the tool.
  Further information: "Turning operation with FreeTurn tools", Page 244

	The NC program remains unchanged except for the calling of the FreeTurn cutting edges, see "Example: Turning with a FreeTurn tool", Page 906

Finishing cycle sequence

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control moves the tool to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 If programmed, the tool traverses to the inclination angle that was calculated by the control based on the minimum and maximum inclination angles you have defined.
3 The control finishes the contour of the finished part (contour starting point to contour end point) simultaneously at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Danger of collision!

The control does not perform collision monitoring (DCM). Danger of collision during machining!

- ▶ Run a simulation to verify the sequence and the contour
- ▶ Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

The cycle uses the position of the tool at cycle call as the cycle starting position. Incorrect pre-positioning can cause contour damage. There is a danger of collision!

- ▶ Move the tool to a safe position in the X and Z axes.

NOTICE

Danger of collision!

If the contour ends too closely at the fixture, a collision between tool and fixture might occur during machining.

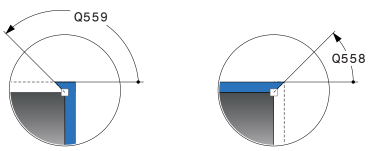
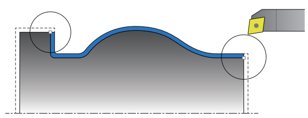
- ▶ When clamping, take both the tool angle of inclination and the departure movement into account

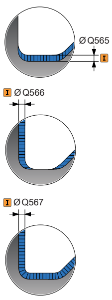
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- Based on the programmed parameters, the control calculates only **one** collision-free path.
- Software limit switches limit the possible inclination angle **Q556** and **Q557**. If, in **Editor** in the **Simulation** the switch for the software end switches is deactivated, then the simulation may deviate from the later machining operation.
- The cycle calculates a collision-free path. For this purpose, it only uses the 2-D contour of the tool holder without considering the Y axis depth.

Notes on programming

- Before programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- Move the tool to a safe position before the cycle call.
- The cycle requires a radius compensation (**RL**/**RR**) in its contour description.
- Prior to the cycle call, you must program **FUNCTION TCPM**. In **FUNCTION TCMP**, HEIDENHAIN recommends programming the tool reference point **REFPNT TIP-CENTER**.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Please note: The smaller the resolution in cycle parameter **Q555** is, the easier will it be to find a solution even in complex situations. The drawback is that the calculation will take more time.
- For determining the inclination angle, the cycle requires the definition of a tool holder. For this purpose, assign a tool holder to the tool in the **KINEMATIC** column of the tool table.
- Please note that cycle parameters **Q565** (Finishing allowance in diameter) and **Q566** (Finishing allowance in Z) cannot be combined with **Q567** (Finishing allowance of contour)!

Cycle parameters

Help graphic	Parameter
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	**Q558 Extensn. angle at contour start?** Angle in the WPL-CS, by which the cycle extends the contour up to the workpiece blank at the programmed starting point. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	**Q559 Extension angle at contour end?** Angle in WPL CS by which the cycle extends the contour at the programmed end point up to the workpiece blank. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q556 Minimum angle of inclination? Smallest possible permitted angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q557 Maximum angle of inclination? Largest possible angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q555 Stepping angle for calculation? Cutting width for the calculation of possible solutions Input: **0.5...9.99**

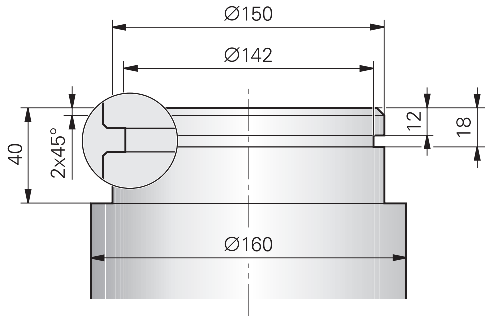
Help graphic	Parameter
	Q537 Inclin. angle (0=N/1=J/2=S/3=E)? Define whether an inclination angle is active: **0**: No inclination angle active **1**: Inclination angle active **2**: Inclination angle at contour start active **3**: Inclination angle at contour end active Input: **0**, **1**, **2**, **3**
	Q538 Inclin. angle at contour start? Inclination angle at the beginning of the programmed contour (WPL-CS) Input: **-180...+180**
 	**Q539 Inclinatn. angle at contour end?** Inclination angle at the end of the programmed contour (WPL-CS) Input: **-180...+180**
	Q565 Finishing allowance in diameter Diameter oversize that remains on the contour after finishing. This value has an incremental effect. Input: **-9...99.999**
	Q566 Finishing allowance in Z? Oversize on the defined contour in the axial direction that remains on the contour after finishing. This value has an incremental effect. Input: **-9...99.999**
	Q567 Finishing allowance of contour? Contour-parallel oversize on the defined contour that remains after finishing. This value has an incremental effect. Input: **-9...99.999**

Example

```
11 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q499=+0                 ;REVERSE CONTOUR ~
    Q558=+0                 ;EXT:ANGLE CONT.START ~
    Q559=+90                ;CONTOUR END EXT ANGL ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q556=-30                ;MIN. INCLINAT. ANGLE ~
    Q557=+30                ;MAX. INCLINAT. ANGLE ~
    Q555=+7                 ;STEPPING ANGLE ~
    Q537=+0                 ;INCID. ANGLE ACTIVE ~
    Q538=+0                 ;INCLIN. ANGLE START ~
    Q539=+0                 ;INCLINATN. ANGLE END ~
    Q565=+0                 ;FINISHING ALLOW. D. ~
    Q566=+0                 ;FINISHING ALLOW. Z ~
    Q567=+0                 ;FINISH. ALLOW. CONT.
12 L  X+58  Y+0 FMAX M303
13 L  Z+50 FMAX
14 CYCL CALL
```

15.4.35 Programming examples

Example: Shoulder with recess



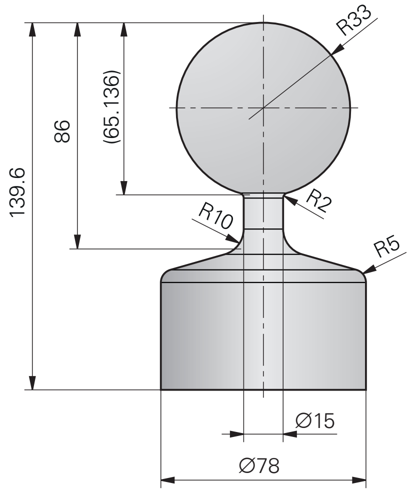
0 BEGIN PGM 9 MM	
1 BLK FORM CYLINDER Z R80 L60	
2 TOOL CALL 301	; Tool call
3 M140 MB MAX	; Retract the tool
4 FUNCTION MODE TURN	; Activate turning mode
5 FUNCTION TURNDATA SPIN VCONST:ON VC:150	; Constant cutting speed
6 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532=+750 ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
7 M136	; Feed rate in mm/rev.
8 L X+165 Y+0 R0 FMAX	; Approach starting point in the plane
9 L Z+2 R0 FMAX M304	; Safety clearance, turning spindle on
10 CYCL DEF 812 SHOULDER, LONG. EXT. ~	
Q215=+0 ;MACHINING OPERATION ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q491=+160 ;DIAMETER AT CONTOUR START ~	
Q492=+0 ;CONTOUR START IN Z ~	
Q493=+150 ;DIAMETER AT CONTOUR END ~	
Q494=-40 ;CONTOUR END IN Z ~	
Q495=+0 ;ANGLE OF CIRCUM. SURFACE ~	
Q501=+1 ;TYPE OF STARTING ELEMENT ~	
Q502=+2 ;SIZE OF STARTING ELEMENT ~	
Q500=+1 ;RADIUS OF CONTOUR EDGE ~	
Q496=+0 ;ANGLE OF FACE ~	
Q503=+1 ;TYPE OF END ELEMENT ~	

Q504=+2 ;SIZE OF END ELEMENT ~	
Q463=+2.5 ;MAX. CUTTING DEPTH ~	
Q478=+0.25 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q484=+0.2 ;OVERSIZE IN Z ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q506=+0 ;CONTOUR SMOOTHING	
11 CYCL CALL	; Cycle call
12 M305	; Turning spindle off
13 TOOL CALL 307	; Tool call
14 M140 MB MAX	; Retract the tool
15 FUNCTION TURNDATA SPIN VCONST:ON VC:100	; Constant cutting speed
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532=+750 ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+0 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
17 L X+165 Y+0 R0 FMAX	; Approach starting point in the plane
18 L Z+2 R0 FMAX M304	; Safety clearance, turning spindle on
19 CYCL DEF 862 EXPND. RECESS, RADL. ~	
Q215=+0 ;MACHINING OPERATION ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q491=+150 ;DIAMETER AT CONTOUR START ~	
Q492=-12 ;CONTOUR START IN Z ~	
Q493=+142 ;DIAMETER AT CONTOUR END ~	
Q494=-18 ;CONTOUR END IN Z ~	
Q495=+0 ;ANGLE OF SIDE ~	
Q501=+1 ;TYPE OF STARTING ELEMENT ~	
Q502=+1 ;SIZE OF STARTING ELEMENT ~	
Q500=+0 ;RADIUS OF CONTOUR EDGE ~	
Q496=+0 ;ANGLE OF SECOND SIDE ~	
Q503=+1 ;TYPE OF END ELEMENT ~	
Q504=+1 ;SIZE OF END ELEMENT ~	
Q478=+0.3 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q484=+0.2 ;OVERSIZE IN Z ~	
Q505=+0.15 ;FINISHING FEED RATE ~	
Q463=+0 ;LIMIT TO DEPTH ~	

Q510=+0.8 ;RECESSING OVERLAP ~	
Q511=+80 ;FEED RATE FACTOR ~	
Q462=+0 ;RETRACTION MODE ~	
Q211=+3 ;DWELL TIME IN REVS ~	
Q562=+1 ;MULTIPLE PLUNGING	
20 CYCL CALL M8	; Cycle call
21 M305	; Turning spindle off
22 M137	; Feed rate in mm/minute
23 M140 MB MAX	; Retract the tool
24 FUNCTION MODE MILL	; Activate milling mode
25 M30	; End of program
26 END PGM 9 MM	

Example: Simultaneous turning

The following NC program uses Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** and Cycle **883 TURNING SIMULTANEOUS FINISHING**.



Program sequence

- Call the tool (e.g., TURN_ROUGH)
- Activate turning mode
- Pre-position
- Select the contours by using **SEL CONTOUR**
- Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Call the cycle
- Tool call (e.g., TURN_FINISH)
- Activate turning mode
- Cycle **883 TURNING SIMULTANEOUS FINISHING**
- Call the cycle
- End of program

0 BEGIN PGM 1341941_1 MM	
1 BLK FORM ROTATION Z DIM_D FILE "1341941_blank.H"	
2 FUNCTION MODE TURN	; Activate turning mode
3 TOOL CALL "TURN_ROUGH"	; Tool call
4 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+1 ;ANGLE OF INCIDENCE ~	
Q532=MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	

5 FUNCTION TURNDATA SPIN VCONST: ON VC:400 SMAX800	; Constant surface speed
6 M145	; Reset the tool offset
7 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	; Activate TCPM
8 L X+120 Y+0 R0 FMAX	; Pre-position
9 L Z+20 R0 FMAX M303	
10 FUNCTION TURNDATA BLANK "1341941_blank.H"	; Workpiece blank update
11 SEL CONTOUR "1341941_finish.h"	; Define the contour
12 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=-90 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q478=+0.3 ;ROUGHING FEED RATE ~	
Q488=+0.3 ;PLUNGING FEED RATE ~	
Q556=-80 ;MIN. INCLINAT. ANGLE ~	
Q557=+90 ;MAX. INCLINAT. ANGLE ~	
Q567=+0.4 ;FINISH. ALLOW. CONT. ~	
Q519=+2 ;INFEED ~	
Q463=+2.5 ;MAX. CUTTING DEPTH ~	
Q590=+1 ;MACHINING MODE ~	
Q591=+0 ;MACHINING SEQUENCE ~	
Q389=+0 ;UNI.- BIDIRECTIONAL	
13 CYCL CALL	; Cycle call
14 M305	
15 TOOL CALL "TURN_FINISH"	; Tool call
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+1 ;ANGLE OF INCIDENCE ~	
Q532=MAX ;FEED RATE ~	
Q533=+1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
17 FUNCTION TURNDATA SPIN VCONST: ON VC:400 SMAX800	; Constant surface speed
18 M145	; Reset the tool offset
19 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	; Activate TCPM
20 L X+120 Y+0 R0 FMAX	

21 L Z+20 R0 FMAX M303	
22 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=-90 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q556=-80 ;MIN. INCLINAT. ANGLE ~	
Q557=+90 ;MAX. INCLINAT. ANGLE ~	
Q555=+1 ;STEPPING ANGLE ~	
Q537=+0 ;INCID. ANGLE ACTIVE ~	
Q538=+0 ;INCLIN. ANGLE START ~	
Q539=+0 ;INCLINATN. ANGLE END ~	
Q565=+0 ;FINISHING ALLOW. D. ~	
Q566=+0 ;FINISHING ALLOW. Z ~	
Q567=+0 ;FINISH. ALLOW. CONT.	
23 CYCL CALL	; Cycle call
24 M305	
25 FUNCTION TURNDATA BLANK OFF	; Deactivate workpiece blank update
26 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
27 FUNCTION MODE MILL	; Activate milling mode
28 TOOL CALL 0 Z	
29 PLANE RESET TURN FMAX	
30 M30	; End of program
31 END PGM 1341941_1 MM	

NC program 1341941_blank.h

```
0  BEGIN PGM 1341941_BLANK MM
1  L  X+0  Z+0.4
2  L  X+80
3  L  Z-139.6
4  L  X+0
5  L  Z+0.4
6  END PGM 1341941_BLANK MM
```

NC program 1341941_finish.h

```
0  BEGIN PGM 1341941_FINISH MM
1  L  X+0  Z+0 RR
2  CR  Z-65.136  X+15 R+33 DR+
3  RND R2
4  L  Z-86
5  RND R10
6  L  X+78 Z-95
7  RND R5
8  L  Z-100
9  END PGM 1341941_FINISH MM
```

Example: Turning with a FreeTurn tool

Cycles **882 SIMULTANEOUS ROUGHING FOR TURNING** and **883 TURNING SIMULTANEOUS FINISHING** are used in the following NC program.

Program sequence:

- Activate turning mode
- Call FreeTurn tool with second cutting edge
- Adjust the coordinate system with cycle **800 ADJUST XZ SYSTEM**
- Move to safe position
- Call cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Call FreeTurn tool with second cutting edge
- Move to safe position
- Call cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Move to safe position
- Call cycle **883 TURNING SIMULTANEOUS FINISHING**
- Reset active transformation with the PC program **RESET.h**

0 BEGIN PGM FREETURN MM	
1 FUNCTION MODE TURN "AC_TURN"	; Activate turning mode
2 PRESET SELECT #16	
3 BLK FORM CYLINDER Z D100 L101 DIST+1	
4 FUNCTION TURNDATA BLANK LBL 1	; Activate blank form update
5 TOOL CALL 145.0	; Call FreeTurn tool with first edge
6 M136	
7 FUNCTION TURNDATA SPIN VCONST:ON VC:250	; Constant cutting speed
8 L Z+50 R0 FMAX M303	
9 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+90 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
10 CYCL DEF 14.0 CONTOUR	
11 CYCL DEF 14.1 KONTURLABEL2	
12 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=+0 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q478=+0.3 ;ROUGHING FEED RATE ~	
Q488=+0.3 ;PLUNGING FEED RATE ~	

Q556=+30 ;MIN. INCLINAT. ANGLE ~	
Q557=+160 ;MAX. INCLINAT. ANGLE ~	
Q567=+0.3 ;FINISH. ALLOW. CONT. ~	
Q519=+2 ;INFEED ~	
Q463=+2 ;MAX. CUTTING DEPTH ~	
Q590=+5 ;MACHINING MODE ~	
Q591=+1 ;MACHINING SEQUENCE ~	
Q389=+0 ;UNI.- BIDIRECTIONAL	
13 L X+105 Y+0 R0 FMAX	
14 L Z+2 R0 FMAX M99	
15 TOOL CALL 145.1	; Call FreeTurn tool with second cutting edge
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+90 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
17 Q519 = 1	; Reduce infeed to 1
18 L X+105 Y+0 R0 FMAX	; Approach starting point
19 L Z+2 R0 FMAX M99	; Call cycle
20 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=+0 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q556=+30 ;MIN. INCLINAT. ANGLE ~	
Q557=+160 ;MAX. INCLINAT. ANGLE ~	
Q555=+5 ;STEPPING ANGLE ~	
Q537=+0 ;INCID. ANGLE ACTIVE ~	
Q538=+90 ;INCLIN. ANGLE START ~	
Q539=+0 ;INCLINATN. ANGLE END ~	
Q565=+0 ;FINISHING ALLOW. D. ~	
Q566=+0 ;FINISHING ALLOW. Z ~	
Q567=+0 ;FINISH. ALLOW. CONT.	
21 L X+105 Y+0 R0 FMAX	; Approach starting point
22 L Z+2 R0 FMAX M99	; Call cycle
23 CALL PGM RESET.H	; Call **RESET** program

24 M30	; End of program
25 LBL 1	; Define **LBL 1**
26 L X+100 Z+1	
27 L X+0	
28 L Z-60	
29 L X+100	
30 L Z+1	
31 LBL 0	
32 LBL 2	; Define **LBL 2**
33 L Z+1 X+60 RR	
34 L Z+0	
35 L Z-2 X+70	
36 RND R2	
37 L X+80	
38 RND R2	
39 L Z+0 X+98	
40 RND R2	
41 L Z-10	
42 RND R2	
43 L Z-8 X+89	
44 RND R2	
45 L Z-15 X+60	
46 RND R2	
47 L Z-55	
48 RND R2	
49 L Z-50 X+98	
50 RND R2	
51 L Z-60	
52 LBL 0	
53 END PGM FREETURN MM	

15.5 Cycles for grinding

15.5.1 Overview

Reciprocating stroke

Cycle	Call	Further information
1000 DEFINE RECIP. STROKE (option 156) ■ Define the reciprocating stroke and start it, if applicable	**DEF**-active	Page 911
1001 START RECIP. STROKE (option 156) ■ Start reciprocating stroke	**DEF**-active	Page 914
1002 STOP RECIP. STROKE (option 156) ■ Stop the reciprocating stroke and clear it, if applicable	**DEF**-active	Page 915

Dressing cycles

Cycle	Call	Further information
1010 DRESSING DIAMETER (option 156) ■ Dressing a grinding wheel diameter	**DEF**-active	Page 918
1015 PROFILE DRESSING (option 156) ■ Dressing a defined grinding wheel profile	**DEF**-active	Page 922
1016 DRESSING OF CUP WHEEL (option 156) ■ Dressing a cup wheel	**DEF**-active	Page 926

1017 DRESSING WITH DRESSING ROLL (option 156)
■ Dressing with a dressing roll
  ■ Reciprocating strokes
  ■ Oscillating
  ■ Fine oscillating
	DEF-active	Page 931

1018 RECESSING WITH DRESSING ROLL (option 156)
■ Dressing with a dressing roll
  ■ Recessing
  ■ Multi-recessing
	DEF-active	Page 937

Contour grinding cycles

Cycle	Call	Further information
1021 CYLINDER, SLOW-STROKE GRINDING (option 156) ■ Grinding of cylindrical internal or external contours ■ Multiple circular paths during a reciprocating stroke	**CALL**-active	Page 943
1022 CYLINDER, FAST-STROKE GRINDING (option 156) ■ Grinding of cylindrical internal or external contours ■ Grinding with circular and helix paths, movement superimposed w/ reciprocating stroke, as required	**CALL**-active	Page 951
1025 GRINDING CONTOUR (option 156) ■ Grinding open and closed contours	**CALL**-active	Page 957

Special cycles

Cycle	Call	Further information
1030 **ACTIVATE WHEEL EDGE** (option 156) ■ Activating the desired wheel edge	**DEF**-active	Page 960
1032 **GRINDING WHL LENGTH COMPENSATION** (option 156) ■ Compensation of the length in absolute or incremental values	**DEF**-active	Page 962
1033 **GRINDING WHL RADIUS COMPENSATION** (option 156) ■ Compensation of the radius in absolute or incremental values	**DEF**-active	Page 964

15.5.2 General information on jig grinding

General information on jig grinding

Jig grinding means grinding of a 2-D contour. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin. Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Outline: Grinding with a reciprocating stroke

```
0 BEGIN PGM GRIND MM
1 FUNCTION MODE MILL
2 TOOL CALL "GRIND_1" Z S20000
3 CYCL DEF 1000 DEFINE RECIP. STROKE
...
4 CYCL DEF 1001 START RECIP. STROKE
...
5 CYCL DEF 14 CONTOUR
...
6 CYCL DEF 1025 GRINDING CONTOUR
...
7 CYCL CALL
8 CYCL DEF 1002 STOP RECIP. STROKE
...
9 END PGM GRIND MM
```

15.5.3 Cycle 1000 DEFINE RECIP. STROKE (option 156)

ISO programming
G1000

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1000 DEFINE RECIP. STROKE** to define a reciprocating stroke in the tool axis and start reciprocating. This movement is executed as a superimposed movement. Thus, it is possible to execute any positioning block in parallel to the reciprocating stroke, even in the axis that is reciprocating. Once you started the reciprocating stroke, you can call a contour and start grinding.

- If you set **Q1004** to **0**, no reciprocating stroke will take place. In this case, you only define the cycle. If required, call Cycle **1001 START RECIP. STROKE** later to start the reciprocating stroke
- If you set **Q1004** to **1**, the reciprocating stroke starts at the current position. Depending on the setting in **Q1002**, the control will start reciprocating the tool in the positive or negative direction first. This reciprocation movement will be superimposed on the programmed movements (X, Y, Z)

The following cycles can be called in combination with the reciprocating stroke:

- Cycle **24 SIDE FINISHING**
- Cycle **25 CONTOUR TRAIN**
- Cycles **25x POCKETS/STUDS/SLOTS**
- Cycle **276 THREE-D CONT. TRAIN**
- Cycle **274 OCM FINISHING SIDE**
- Cycle **1025 GRINDING CONTOUR**

> - The control does not support mid-program startup while the reciprocating stroke is active.
> - As long as the reciprocating stroke is active in the started NC program, you cannot switch to the **MDI** application in **Manual** operating mode.

Notes

Refer to your machine manual!
The overrides for the reciprocation movements can be changed by the machine manufacturer.

NOTICE

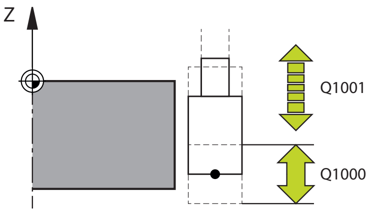
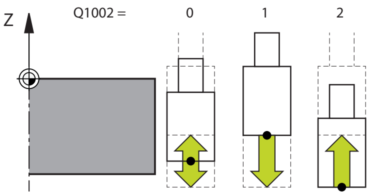
Danger of collision!

Collision monitoring (DCM) is not active during reciprocation movements. This means that movements that might cause collisions will not be prevented. There is a danger of collision!

- Verify the NC program by carefully executing it block by block

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1000** is DEF-active.
- The simulation of the superimposed movement can be seen in **Program Run** mode and in **Single Block** mode.
- Stop the reciprocating movement when you no longer need it. To do so, use **M30** or Cycle **1002 STOP RECIP. STROKE**. **STOP** or **M0** will not stop the reciprocating stroke.
- Reciprocating strokes can also be started in a tilted working plane. While the reciprocating stroke is active, however, you cannot change the orientation of the plane.
- You can also use a milling cutter with the superimposed reciprocating movement.

Cycle parameters

Help graphic	Parameter
	**Q1000 Length of reciprocating stroke?** Length of the reciprocating movement, parallel to the active tool axis Input: **0...9999.9999**
	Q1001 Feed rate for reciprocation? Speed of the reciprocating stroke in mm/min Input: **0...999999**
	**Q1002 Type of reciprocation?** Definition of the start position. The direction of the first reciprocating stroke arises from this. **0**: The current position is the middle of the stroke. The control first offsets the grinding tool by half the stroke in the negative direction and then continues the reciprocating movement in the positive direction **-1**: The current position is the upper limit of the stroke. During the first stroke, the control offsets the grinding tool in the negative direction. **+1**: The current position is the lower limit of the stroke. For the first stroke, the control offsets the grinding tool in the positive direction Input: **-1**, **0**, **+1**
	Q1004 Start reciprocating stroke? Definition of the effect of this cycle: **0**: The reciprocating stroke is merely defined and may be started at a later time **+1**: The reciprocating stroke is defined and started at the current position Input: **0**, **1**

Example

```
11 CYCL DEF 1000 DEFINE RECIP. STROKE ~
    Q1000=+0              ;RECIPROCATING STROKE ~
    Q1001=+999            ;RECIP. FEED RATE ~
    Q1002=+1              ;RECIPROCATION TYPE ~
    Q1004=+0              ;START RECIP. STROKE
```

15.5.4 Cycle 1001 START RECIP. STROKE (option 156)

ISO programming
G1001

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.

Cycle **1001 START RECIP. STROKE** starts a previously defined or stropped reciprocation movement. In an ongoing movement, this cycle has no effect.

Notes

⚙	Refer to your machine manual! The overrides for the reciprocation movements can be changed by the machine manufacturer.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1001** is DEF-active.
- If you did not define a reciprocating stroke with Cycle **1000 DEFINE RECIP. STROKE**, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	Cycle **1001** does not have a cycle parameter. Close cycle input with the **END** key.

Example

```
11 CYCL DEF 1001 START RECIP. STROKE
```

15.5.5 Cycle 1002 STOP RECIP. STROKE (option 156)

ISO programming
G1002

Application

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

Cycle **1002 STOP RECIP. STROKE** stops the reciprocation movement. Depending on the setting in **Q1010**, the tool will stop immediately or traverse to its starting position.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1002** is DEF-active.

Note on programming

- Stopping the movement at the current position (**Q1010**=1) is allowed only if you simultaneously clear the definition of the reciprocating stroke (**Q1005**=1).

Cycle parameters

Help graphic	Parameter
	Q1005 Clear reciprocating stroke? Definition of the effect of this cycle: **0**: The reciprocating stroke is merely stopped and may be started again at a later time **+1**: The reciprocating stroke is stopped, and the definition of the reciprocating stroke from cycle **1000** is cleared Input: **0**, **1**
	Q1010 Stop reciproc. immediately (1)? Definition of the stopping position of the grinding tool: **0**: The stopping position is the same as the starting position **+1**: The stopping position is the same as the current position Input: **0**, **1**

Example

```
11 CYCL DEF 1002 STOP RECIP. STROKE ~
    Q1005=+0                    ;CLEAR RECIP. STROKE ~
    Q1010=+0                    ;RECIP.STROKE STOPPOS
```

15.5.6 General information on the dressing cycles

Fundamentals

> Refer to your machine manual.
> For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.

The term "dressing" refers to the sharpening or trueing up of a grinding tool inside the machine. During dressing, the dresser machines the grinding wheel. Thus, in dressing, the grinding tool is the workpiece.

The dressing operation removes material from the grinding wheel and may cause wear of the dressing tool. The material removal and wear lead to changed tool data that need to be compensated for after dressing.

The following dressing cycles are available:

- **1010 DRESSING DIAMETER**, Page 918
- **1015 PROFILE DRESSING**, Page 922
- **1016 DRESSING OF CUP WHEEL**, Page 926
- **1017 DRESSING WITH DRESSING ROLL**, Page 931
- **1018 RECESSING WITH DRESSING ROLL**, Page 937

In dressing, the workpiece datum is located on an edge of the grinding wheel. Select the respective edge by using Cycle **1030 ACTIVATE WHEEL EDGE**.

Identify dressing operations in your NC program with **FUNCTION DRESS BEGIN / END**. When you activate **FUNCTION DRESS BEGIN**, the grinding wheel is redefined as the workpiece and the dressing tool as the tool. This might result in the axes moving in the opposite direction. When you terminate the dressing mode with **FUNCTION DRESS END**, the grinding wheel is redefined as the tool.

Further information: "Dressing", Page 251

Structure of an NC program for dressing:

- Activate milling mode
- Call grinding wheel
- Move the tool to be dressed to a position near the dressing tool
- Activate dressing mode; select the kinematic model if necessary
- Activate wheel edge
- Call dressing tool; no mechanical tool change
- Call the cycle for dressing the diameter
- Deactivate dressing mode

```
0 BEGIN PGM GRIND MM
1 FUNCTION MODE MILL
2 TOOL CALL "GRIND_1" Z S20000
3 L X... Y... Z...
4 FUNCTION DRESS BEGIN
5 CYCL DEF 1030 ACTIVATE WHEEL EDGE
...
6 TOOL CALL "DRESS_1"
7 CYCL DEF 1010 DRESSING DIAMETER
...
8 FUNCTION DRESS END
9 END PGM GRIND MM
```

- The control does not support mid-program startup while dressing is active. If you jump to the first NC block after dressing using mid-program startup, the control will move the tool to the last position approached during dressing.

Notes

- If you interrupt a dressing infeed movement, the last infeed will not be considered. If applicable, the dressing tool executes the first infeed or part of it without removing material if the dressing cycle is called again.
- Not all grinding tools require dressing. Comply with the information provided by your tool manufacturer.
- Please note that the switchover to dressing mode might have been programmed into the cycle sequence already by the machine manufacturer.

  Further information: "Dressing", Page 251

15.5.7 Cycle 1010 DRESSING DIAMETER (option 156)

ISO programming
G1010

Application

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

Cycle **1010 DRESSING DIAMETER** allows you to dress the outside diameter of your grinding wheel. Depending on the strategy, the control causes movements based on the wheel geometry. If the dressing strategy in **Q1016** was set to 1 or 2, the path of the tool to the starting point is not along the grinding wheel, but via a retract path. The control does not apply tool radius compensation in the dressing cycle.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	1, 3, 5, 7	not supported

If you work with the dressing roll tool type, then only the grinding pin is permitted.

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 960

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

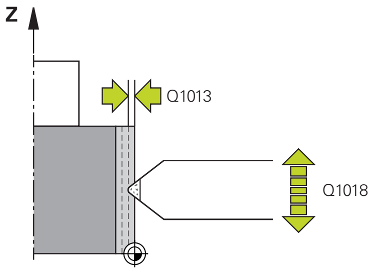
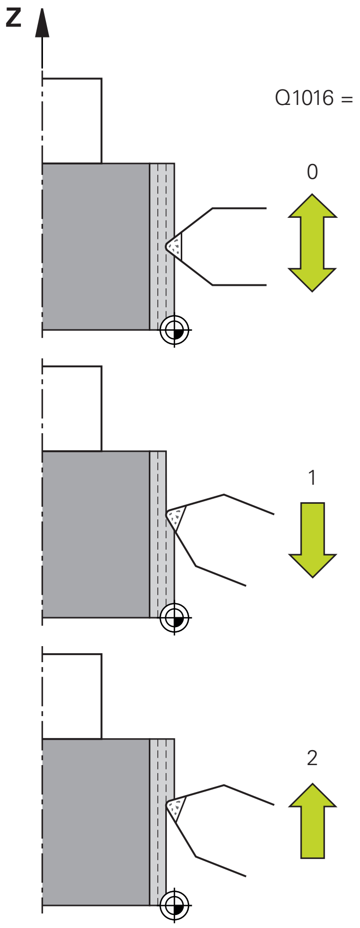
- Cycle **1010** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- The cycle supports dressing with a dressing role.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

  Further information: "Dressing", Page 251

Information about dressing with a dressing role

- For the dressing tool, you must define the dressing role **TYPE**.
- For the dressing role, you must define a width: **CUTWIDTH**. The control takes the width into account during the dressing process.
- For dressing with a dressing role, only the dressing strategy **Q1016=0** is allowed.

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	**Q1016 Dressing strategy (0-2)?** Definition of the traversing movement during dressing: **0**: Reciprocating; dressing occurs in both directions **1**: Pulling; dressing occurs along the grinding wheel solely towards the active wheel edge **2**: Pushing; dressing occurs along the grinding wheel solely away from the active wheel edge Input: **0**, **1**, **2**
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1010 DRESSING DIAMETER ~
    Q1013=+0              ;DRESSING AMOUNT ~
    Q1018=+100            ;DRESSING FEED RATE ~
    Q1016=+1              ;DRESSING STRATEGY ~
    Q1019=+1              ;NUMBER INFEEDS ~
    Q1020=+0              ;IDLE STROKES ~
    Q1022=+0              ;COUNTER FOR DRESSING ~
    Q330=-1               ;TOOL ~
    Q1011=+0              ;FACTOR VC
```

15.5.8 Cycle 1015 PROFILE DRESSING (option 156)

ISO programming
G1015

Application

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

Use Cycle **1015 PROFILE DRESSING** to dress a defined profile of your grinding wheel. The profile must be defined in a separate NC program. This cycle is based on the grinding pin tool type. The starting point and end point of the profile must be identical (closed path) and are located at a corresponding position on the selected wheel edge. Define the return path to the starting point in your profile program. You must program the NC program in the ZX plane. Depending on the profile program, the control either does or does not use tool radius compensation. The activated wheel edge is used as the preset.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 960

Cycle sequence

1 The control positions the dressing tool at the starting position with **FMAX**. The distance of the starting position from the datum is equal to the retraction values of the grinding wheel. The retraction values are relative to the active grinding edge.
2 The control offsets the datum to the extent of the dressing value and executes the profile program. This process repeats itself depending on the definition of **NUMBER INFEEDS Q1019**.
3 The control executes the profile program to the extent of the dressing value. If have programmed **NUMBER INFEEDS Q1019**, the infeeds repeat themselves. For every infeed, the dressing tool moves to the extent of the dressing value **Q1013**.
4 The profile program is repeated without infeed in accordance with **IDLE STROKES Q1020**.
5 The motion ends in the starting position.

	■ The datum of the workpiece system lies on the active wheel edge.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

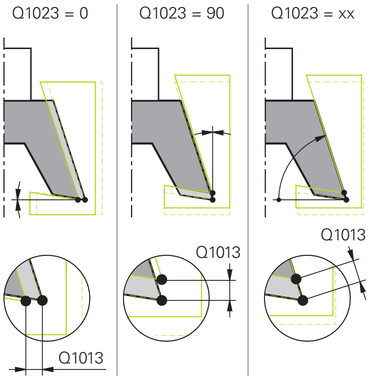
- Cycle **1015** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

  Further information: "Dressing", Page 251

Note on programming

- The angle of infeed must be selected in a way that the programmed profile always remains within the grinding wheel edge. If this condition is not met, then the dimensional accuracy of the grinding wheel is lost.

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1023 Infeed angle of profile program? Angle at which the profile of the program is moved into the grinding wheel. **0**: Infeed only at the diameter in the X axis of the dressing kinematic model **+90**: Infeed only in the Z axis of the dressing kinematic model Input: **0...90**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	Q1000 Name of the profile program? Enter the path and name of the NC program that will be used for the profile of the grinding wheel during the dressing process. Alternatively, select the profile program via name option in the action bar. Input: Max. **255** characters
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**

Help graphic	Parameter
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1015 PROFILE DRESSING ~
    Q1013=+0                ;DRESSING AMOUNT ~
    Q1023=+0                ;ANGLE OF INFEED ~
    Q1018=+100              ;DRESSING FEED RATE ~
    QS1000=""               ;PROFILE PROGRAM ~
    Q1019=+1                ;NUMBER INFEEDS ~
    Q1020=+0                ;IDLE STROKES ~
    Q1022=+0                ;COUNTER FOR DRESSING ~
    Q330=-1                 ;TOOL ~
    Q1011=+0                ;FACTOR VC
```

15.5.9 Cycle 1016 DRESSING OF CUP WHEEL (option 156)

ISO programming
G1016

Application

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

Use Cycle **1016 DRESSING OF CUP WHEEL** to dress the front face of a cup wheel. The activated wheel edge is used as the reference.

Depending on the strategy, the control causes movements based on the wheel geometry. If the dressing strategy in **Q1016** was set to **1** or **2**, the return of the tool to the starting point is not along the grinding wheel, but via a retract path.

If the Pull-and-Push strategy has been selected in dressing mode, the control will apply radius compensation. If the Reciprocating strategy has been selected in dressing mode, the control will not apply radius compensation.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
not supported	not supported	2, 6

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 960

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

The angle of inclination between the dressing tool and the cup wheel will not be monitored! There is a danger of collision!

- Make sure to program a dressing tool clearance angle greater than or equal to 0° relative to the front face of the cup wheel
- Verify the NC program by carefully executing it block by block

- Cycle **1016** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- The control saves the counter in the tool table. Its effect is global.
  Further information: "Tool data for the tool types", Page 284
- To enable dressing of the entire cutting edge, it is extended by twice the cutting-edge radius (2 x **RS**) of the dressing tool. Here, the minimum permissible radius (**R_MIN**) of the grinding wheel must not be undershot, otherwise the control interrupts the operation with an error message.
- In this cycle, the radius of the tool shank is not monitored.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

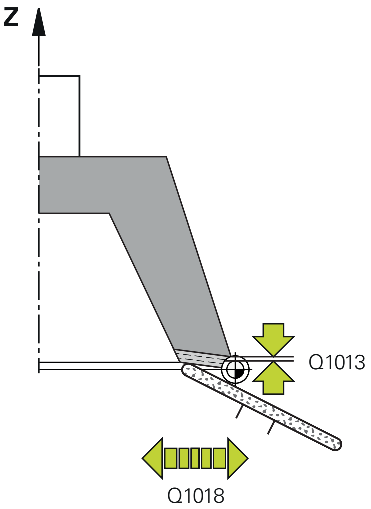
  Further information: "Simplified dressing with a macro", Page 252

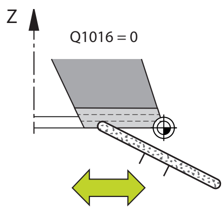
Notes on programming

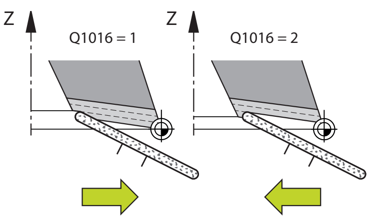
- This cycle is permitted only for use with the cup wheel tool type. If you defined a different tool type, the control will display an error message.
- The strategy in **Q1016** = 0 (Reciprocating) is only possible for a straight front face angle (**HWA** = 0).

Cycle parameters

Help graphic | **Parameter**







Q1013 Dressing amount?

Value used by the control for the dressing infeed.

Input: **0...9.9999**

Q1018 Feed rate for dressing?

Feed rate during the dressing procedure

Input: **0...99999**

Q1016 Dressing strategy (0-2)?

Definition of the traversing movement during dressing:

0: Reciprocating; dressing occurs in both directions

1: Pulling; dressing occurs along the grinding wheel solely towards the active wheel edge

2: Pushing; dressing occurs along the grinding wheel solely away from the active wheel edge

Input: **0**, **1**, **2**

Q1019 Number of dressing infeeds?

Number of infeeds of the dressing process

Input: **1...999**

Q1020 Number of idle strokes?

Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed.

Input: **0...99**

Q1022 Dressing after number of calls?

Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager.

0: The control dresses the grinding wheel during every cycle definition in the NC program.

>0: The control dresses the grinding wheel after this number of cycle definitions.

Input: **0...99**

Q330 Tool number or tool name? (optional)

Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar.

-1: Dressing tool has been activated prior to the dressing cycle

Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1016 DRESSING OF CUP WHEEL ~
    Q1013=+0                ;DRESSING AMOUNT ~
    Q1018=+100              ;DRESSING FEED RATE ~
    Q1016=+1                ;DRESSING STRATEGY ~
    Q1019=+1                ;NUMBER INFEEDS ~
    Q1020=+0                ;IDLE STROKES ~
    Q1022=+0                ;COUNTER FOR DRESSING ~
    Q330=-1                 ;TOOL ~
    Q1011=+0                ;FACTOR VC
```

15.5.10 Cycle 1017 DRESSING WITH DRESSING ROLL (option 156)

ISO programming
G1017

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

With cycle **1017 DRESSING WITH DRESSING ROLL**, you can dress the outside diameter of a grinding wheel with a dressing role. Depending on the dressing strategy, the control performs the appropriate movements in accordance with the wheel geometry.

The cycle offers the following dressing strategies:

- Reciprocating: lateral infeed at the reversal points of the reciprocating stroke
- Oscillating: interpolating infeed during a reciprocating stroke
- Fine Oscillating: interpolating infeed during a reciprocating stroke. After each interpolating infeed, a Z movement is performed without infeed in the dressing kinematic model.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 960

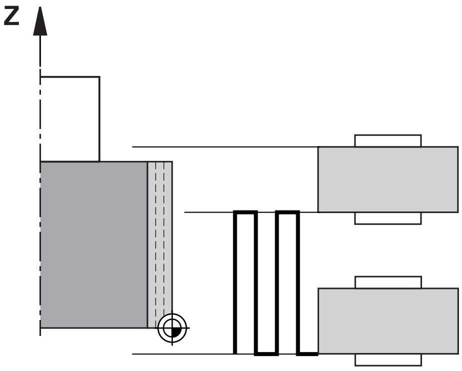
Cycle sequence

1 The control positions the dressing tool at the starting position with **FMAX**.
2 If you have defined a pre-position in **Q1025 PRE-POSITION**, the control approaches the position at **Q253 F PRE-POSITIONING**.
3 The control infeeds based on the dressing strategy.
   Further information: "Dressing strategies", Page 932
4 After defining **IDLE STROKES** in **Q1020**, the control performs them after the last infeed.
5 The control moves to the starting position with **FMAX**.

Dressing strategies

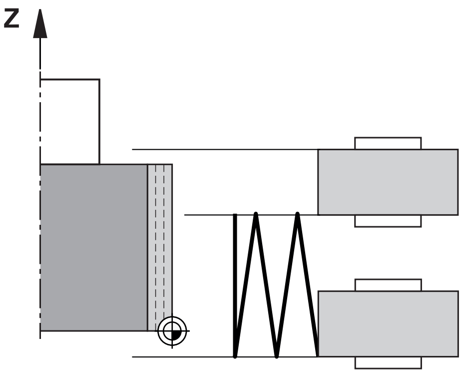
> Depending on **Q1026 WEAR FACTOR**, the control divides the dressing value between the grinding wheel and the dressing roll.

Reciprocating (**Q1024=0**)



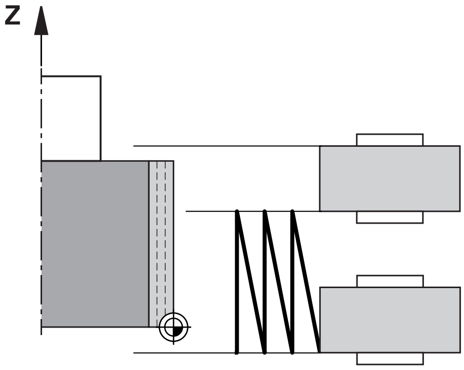
1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The **DRESSING AMOUNT Q1013** is infed on the diameter at the **DRESSING FEED RATE Q1018**.
3 The control moves the dressing tool along the grinding wheel to the next reversal point of the reciprocating movement.
4 If other dressing infeeding is required, the control repeats processes 1 to 2 until the dressing process is complete.

Oscillating (**Q1024=1**)



1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The control infeeds the **DRESSING AMOUNT Q1013** on the diameter. Infeeding is performed with interpolation at the dressing feed rate **Q1018** with the reciprocating stroke up to the next reversal point.
3 If there are more dressing infeed runs, then processes 1 to 2 are repeated until the dressing process is complete.
4 The control then retracts the tool without infeed in the Z axis of the dressing kinematic model to the other reversal point of the reciprocating movement.

Fine oscillating (**Q1024=2**)



1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The control infeeds the **DRESSING AMOUNT Q1013** on the diameter. Infeeding is performed with interpolation at the dressing feed rate **Q1018** with the reciprocating stroke up to the next reversal point.
3 The control then retracts the tool to the other reversal point of the reciprocating movement without an infeed cut.
4 If there is more infeeding, then processes 1 to 3 are repeated until the dressing procedure is complete.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

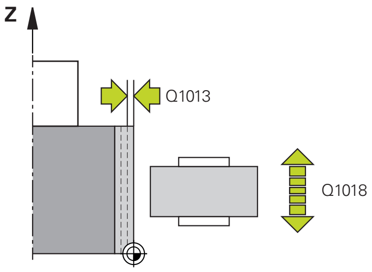
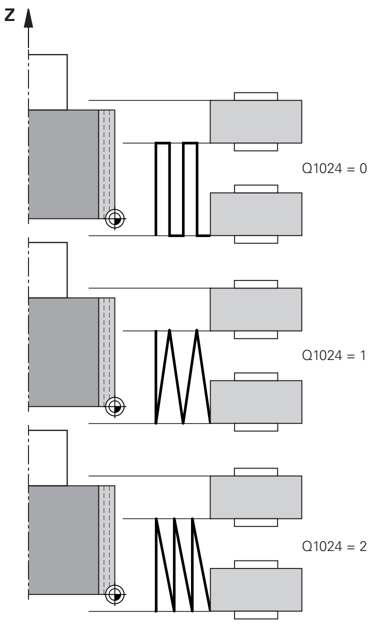
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

- Cycle **1017** is DEF-active.
- No coordinate conversion cycles are permitted in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, then the control performs the dressing process only after reaching the defined counter from the tool management function. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019
- At the end of each infeed run, the control updates the tool data for the grinding tool and dressing tool.
- For the reversal points of the reciprocating movement, the control takes into account the retraction values **AA** and **AI** from the tool management function. The width of the dressing roll must be less than the width of the dressing wheel, including the retraction values.
- The control does not apply tool radius compensation in the dressing cycle.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

  Further information: "Simplified dressing with a macro", Page 252

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	**Q1024 Dressing strategy (0-2)?** Strategy during dressing with a dressing roll; **0**: Reciprocating; infeeding to the reversal points of the reciprocating motion. After the infeeding runs, the control executes a movement just in the Z axis within the dressing kinematic model. **1**: Oscillating; interpolated infeed during a reciprocating movement **2**: Fine oscillating; interpolated during a reciprocating movement. After every interpolated infeed run, the control executes a movement solely in the Z axis in the dressing kinematic model. Input: **0**, **1**, **2**
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1025 Distance for pre-positioning? Distance between the grinding wheel and the dressing role during pre-positioning Input: **0...9.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min. while approaching the pre-position Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q1026 Wear on dressing tool? Factor of the dressing value in order to define the wear on the dressing roll: **0**: The full dressing value is removed on the grinding wheel. **>0**: The factor is multiplied by the dressing value. The control takes the calculated value into account and assumes that this value will be lost during dressing due to wear on the dressing roll. The remaining dressing value is dressed on the grinding wheel. Input: **0...+0.99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1017 DRESSING WITH DRESSING ROLL ~
    Q1013=+0                ;DRESSING AMOUNT ~
    Q1018=+100              ;DRESSING FEED RATE ~
    Q1024=+0                ;DRESSING STRATEGY ~
    Q1019=+1                ;NUMBER INFEEDS ~
    Q1020=+0                ;IDLE STROKES ~
    Q1025=+5                ;PRE-POSITION DIST. ~
    Q253=+1000              ;F PRE-POSITIONING ~
    Q1026=+0                ;WEAR FACTOR ~
    Q1022=+2                ;COUNTER FOR DRESSING ~
    Q330=-1                 ;TOOL ~
    Q1011=+0                ;FACTOR VC
```

15.5.11 Cycle 1018 RECESSING WITH DRESSING ROLL (option 156)

ISO programming
G1018

Application

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

With cycle **1018 RECESSING WITH DRESSING ROLL**, you can dress the outside diameter of a grinding wheel via recessing with dressing role. Depending on the dressing strategy, the control executes one or more recessing movements.

The cycle offers the following dressing strategies:

- **Recessing:** This strategy performs only linear recessing movements. The width of the dressing roll is larger than the dressing wheel width.
- **Multiple recessing:** This strategy executes linear recessing movements. At the end of the infeed run, the control moves the dressing tool in the Z axis of the dressing kinematic model and infeeds again.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 960

Cycle sequence

Recessing

1 The control positions the dressing roll at the starting position at **FMAX**. At the starting position, the center of the dressing roll matches the middle of the grinding wheel edge. If **CENTER OFFSET Q1028** is programmed, then the control takes this into account when approaching the starting position.
2 The dressing roll approaches the **PRE-POSITION DIST. Q1025** at the feed rate **Q253 F PRE-POSITIONING**.
3 The dressing roll recesses into the grinding wheel with the **DRESSING FEED RATE Q1018** by the **DRESSING AMOUNT Q1013**.
4 If a **DWELL TIME IN REVS Q211** is defined, the control waits the defined amount of time.
5 The control retracts the dressing role with **F PRE-POSITIONING Q253** to the **PRE-POSITION DIST. Q1025**.
6 The control moves to the starting position with **FMAX**.

Multiple recessing

1 The control positions the dressing roll at the starting position with **FMAX**.
2 The dressing role approaches the **PRE-POSITION DIST.VORPOSITION Q1025** at the feed rate **Q253F PRE-POSITIONING**.
3 The dressing roll recesses into the grinding wheel with the **DRESSING FEED RATE Q1018** by the **DRESSING AMOUNT Q1013**.
4 If a **DWELL TIME IN REVS Q211** is defined, then it is executed by the control.
5 At **F PRE-POSITIONING Q253**, the control retracts the dressing roll to the **PRE-POSITION DIST. Q1025**.
6 Based on the **RECESSING OVERLAP Q510**, the control moves the dressing roll to the next recessing position in the Z axis of the dressing kinematic model.
7 The control repeats processes 3 to 6 until the entire grinding wheel is dressed.
8 At **F PRE-POSITIONING Q253**, the control retracts the dressing role to the **PRE-POSITION DIST. Q1025**.
9 The control moves to the starting position at rapid traverse.

> The control calculates the number or required recesses based on the width of the grinding wheel, the width of the dressing roll and the value of the parameter **RECESSING OVERLAP Q510**.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

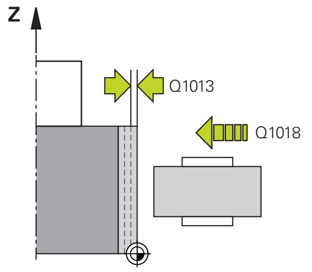
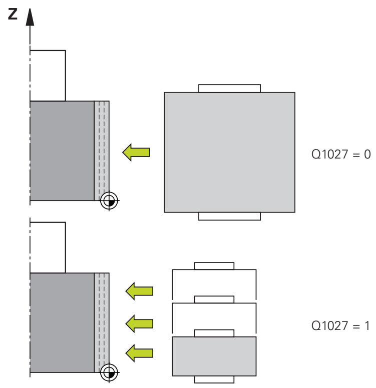
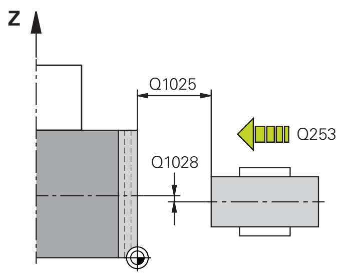
- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

- Cycle **1018** is DEF-active.
- No coordinate transformations are allowed in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation.
- If the width of the dressing roll is less than the width of the grinding wheel, then use the dressing strategy multiple recessing **Q1027=1**.
- If you program a **COUNTER FOR DRESSING Q1022**, then the control performs the dressing process only after reaching the defined counter from the tool management function. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019
- At the end of every infeed run, the control corrects the tool data of the grinding tool and dressing tool.
- The control does not apply tool radius compensation in the dressing cycle.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

  Further information: "Simplified dressing with a macro", Page 252

Cycle parameters

<table>
<tr><th>Help graphic</th><th>Parameter</th></tr>
<tr><td rowspan="2">
</td><td>Q1013 Dressing amount?
Value used by the control for the dressing infeed.
Input: 0...9.9999</td></tr>
<tr><td>Q1018 Feed rate for dressing?
Feed rate during the dressing procedure
Input: 0...99999</td></tr>
<tr><td>
</td><td>Q1027 dressing strategy (0-1)?
Strategy during recessing with a dressing roll:
0: Recessing; the control executes a linear recessing movement. The grinding wheel width is less than the width of the dressing roll.
1: Multiple recessing; the control executes linear recessing movements. After infeeding to the dressing value, the control moves the dressing tool in the Z axis in the dressing kinematic model and infeeds again. The width of the grinding wheel is greater than the width of the dressing roll.
Input: 0, 1</td></tr>
<tr><td rowspan="4">
</td><td>Q1025 Distance for pre-positioning?
Distance between the grinding wheel and the dressing role during pre-positioning
Input: 0...9.9999</td></tr>
<tr><td>Q253 Feed rate for pre-positioning?
Traversing speed of the tool in mm/min. while approaching the pre-position
Input: 0...99999.9999 or FMAX, FAUTO, PREDEF</td></tr>
<tr><td>Q211 Dwell time / 1/min?
Revolutions of the grinding wheel at the end of the recessing cut.
Input: 0...999.99</td></tr>
<tr><td>Q1028 Offset of centers?
Offset of the dressing roll center relative to the grinding wheel center. This offset takes effect in the Z axis of the dressing kinematic model. This value has an incremental effect.
If Q1027 = 1, then the control does not use a center offset.
Input: -999.999...+999.999</td></tr>
</table>

Help graphic	Parameter
	Q510 Overlap factor for recess width? With factor **Q510**, you influence the offset of the dressing roll in the Z axis of the dressing kinematic model. The control multiplies the factor with the value **CUTWIDTH** and offsets the dressing roll between the infeed runs by the calculated value. **1**: For every infeed run, the control recesses with the complete width of the dressing role. **Q510** takes effect only with **Q1027=1**. Input: **0.001...1**
	Q1026 Wear on dressing tool? Factor of the dressing value in order to define the wear on the dressing roll: **0**: The full dressing value is removed on the grinding wheel. **>0**: The factor is multiplied by the dressing value. The control takes the calculated value into account and assumes that this value will be lost during dressing due to wear on the dressing roll. The remaining dressing value is dressed on the grinding wheel. Input: **0...+0.99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1018 RECESSING WITH DRESSING ROLL ~
  Q1013=+1              ;DRESSING AMOUNT ~
  Q1018=+100            ;DRESSING FEED RATE ~
  Q1027=+0              ;DRESSING STRATEGY ~
  Q1025=+5              ;PRE-POSITION DIST. ~
  Q253=+1000            ;F PRE-POSITIONING ~
  Q211=+3               ;DWELL TIME IN REVS ~
  Q1028=+1              ;CENTER OFFSET ~
  Q510=+0.8             ;RECESSING OVERLAP~
  Q1026=+0              ;WEAR FACTOR ~
  Q1022=+2              ;COUNTER FOR DRESSING ~
  Q330=-1               ;TOOL ~
  Q1011=+0              ;FACTOR VC
```

15.5.12 Cycle 1021 CYLINDER, SLOW-STROKE GRINDING (option 156)

ISO programming
G1021

Application

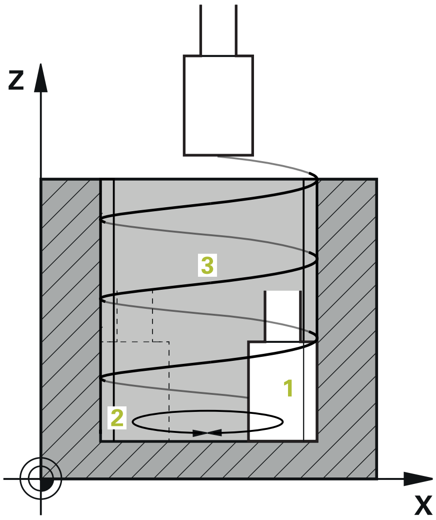
Refer to your machine manual!
This function must be enabled and adapted by the machine manufacturer.

Cycle **1021 CYLINDER, SLOW-STROKE GRINDING** allows you to grind circular pockets or circular studs. The height of the cylinder can be considerably greater than the width of the grinding wheel. Through a reciprocating stroke, the control can machine the complete height of the cylinder. The control executes multiple circular paths during the reciprocating stroke. In this process, the reciprocating stroke and the circular paths overlap to form a helix. This process is equivalent to grinding with a slow stroke.

The lateral infeed cuts occur at the reversal points of the reciprocating stroke along the semi-circle. You can program the feed rate of the reciprocating stroke as the pitch of the helical path relative to the width of the grinding wheel.

You can also completely machine cylinders without overshoot, such as blind holes. This is done by programming idle runs at the reversal points of the reciprocating stroke.

Cycle sequence



1. The control positions the grinding tool above the cylinder, depending on **POCKET POSITION Q367**. The control now traverses the tool to the **CLEARANCE HEIGHT Q260** at rapid traverse.
2. The grinding tool traverses to the **SET-UP CLEARANCE Q200** at **F PRE-POSITIONING Q253**
3. The grinding tool traverses to the starting point in the tool axis. The starting point depends on the **MACHINING DIRECTION Q1031**, upper or lower reversal point of the reciprocating stroke.
4. The cycle starts the reciprocating stroke. At the **GRINDING FEED RATE Q207**, the control moves the grinding tool to the contour.

   Further information: "Feed rate for the reciprocating stroke", Page 945
5. The control delays the reciprocating stroke in the starting position.
6. Depending on **Q1021 ONE-SIDED INFEED**, the control infeeds the grinding tool in a semi-circle around the lateral infeed **Q534** **1**.
7. As needed, the control executes the defined idle runs **2** **Q211** or **Q210**.

   Further information: "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 945
8. The cycle continues the reciprocating movement. The grinding tool follows multiple circular paths. The reciprocating stroke overlays the circular paths in the direction of the tool axis to form a helix. You can influence the pitch of the helical path by the factor **Q1032**.
9. The circular paths **3** repeat themselves until the second reversal point of the reciprocating stroke is reached.
10. The control repeats steps 4 to 7 until the diameter of the finished part **Q223** or the oversize **Q14** is reached.
11. After the last lateral infeed run, the grinding wheel moves the number of programmed idle strokes **Q1020** if applicable.
12. The control stops the reciprocating stroke. The grinding tool leaves the cylinder on a semi-circular path to the safety clearance **Q200**.
13. The grinding tool moves to the **SET-UP CLEARANCE Q200** at the **F PRE-POSITIONING Q253** and then to the **CLEARANCE HEIGHT Q260** at rapid traverse.

- In order for the grinding tool to completely machine the cylinder at the reversal points of the reciprocating stroke, you must define sufficient overshoot or idle runs.
- The length of the reciprocating stroke arises from the **DEPTH Q201**, the **SURFACE OFFSET Q1030** and the wheel width **B**.
- The starting point in the working plane is distant from the **FINISHED PART DIA. Q223** including **OVERSIZE AT START Q368** by the amount of the tool radius and the **SET-UP CLEARANCE Q200**.

Overshoot and idle runs to the reversal points of the reciprocating stroke

Path of the overshoot

Top	Bottom
This distance is defined in the parameter **Q1030 SURFACE OFFSET**.	You must add this distance to the machining depth and then define it in **Q201 DEPTH**.

If no overshoot is possible, such as with a pocket, program multiple idle runs at the reversal points of the reciprocating stroke (**Q210**, **Q211**). Select this number such that, after infeeding (half of a circular path), at least one circular path is traveled on the infed diameter. The number of idle runs is always based on a set feed-rate override of 100%.

- HEIDENHAIN recommends moving with a feed-rate override of 100% or more. A feed-rate override of less than 100% no longer ensures that the cylinder will be completely machined at the reversal points.
- For the definition of idle runs, HEIDENHAIN recommends defining at least a value of 1.5.

Feed rate for the reciprocating stroke

You can define the pitch per helical path (=360°) with the factor **Q1032**. Through this definition, the feed rate in mm or in inches/helical path (= 360°) can be derived for the reciprocating stroke.

The proportion of the **GRINDING FEED RATE Q207** to the feed rate of the reciprocating stroke plays a major role. If you deviate from a feed rate override of 100%, then ensure that the length of the reciprocating stroke during a circular path is less than the width of the grinding wheel.

HEIDENHAIN recommends selecting a factor of at most 0.5.

Notes

The overrides for the reciprocation movements can be changed by the machine manufacturer.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The last lateral infeed may be smaller depending on the input.
- The control does not depict the reciprocating movement in the simulation. The reciprocating movement is depicted in the simulation graphics in the **Program run, single block** and **Program run, full sequence** operating modes.
- You can also execute this cycle with a milling cutter. In the case of a milling cutter, the tooth length **LCUTS** equals the width of the grinding wheel.
- Please note that the cycle takes **M109** into account. The **GRINDING FEED RATE Q207** in the status display during program run in the case of a pocket is therefore smaller than in the case of a stud. The control shows the feed rate of the center point path of the grinding tool, including the reciprocating stroke.

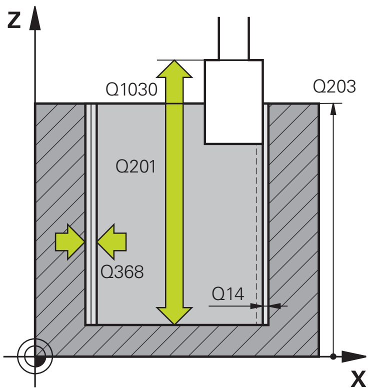
  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

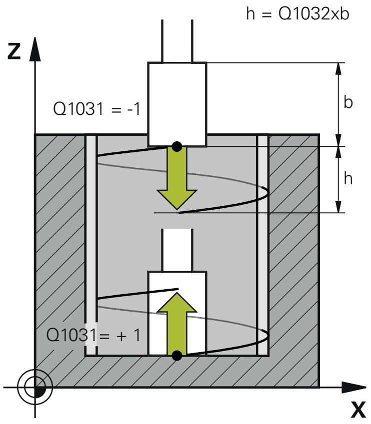
Notes on programming

- The control assumes that the bottom of the cylinder has a floor. For this reason, you can define an overshoot in **Q1030** only at the surface. If you machine a through hole, for example, then you must take into account the lower overshoot in **DEPTH Q201**.

  Further information: "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 945
- If the grinding wheel is wider than **DEPTH Q201** and the **SURFACE OFFSET Q1030**, then the control issues a **No swing stroke** error message. In this case, the resulting reciprocating stroke would be equal to 0.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island Input: **0**, **1**
	Q223 Finished part diameter? Diameter of the fully machined cylinder Input: **0...99999.9999**
	Q368 Side oversize before machining? Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q367 Position of pocket (0/1/2/3/4)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool pos. = Center of figure **1**: Tool pos. = Quadrant transition at 90° **2**: Tool pos. = Quadrant transition at 0° **3**: Tool pos. = Quadrant transition at 270° **4**: Tool pos. = Quadrant transition at 180° Input: **0**, **1**, **2**, **3**, **4**
	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**
	Q1030 Offset to surface? Position of the upper edge of the tool on the surface. The offset serves as the overshoot path on the surface for the reciprocating stroke. This value has an absolute effect. Input: **0...999.999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**

Help graphic	Parameter
	**Q1031 Machining direction?** Definition of the start position. The direction of the first reciprocating stroke arises from this. **−1** or **0**: The starting position is on the surface. The reciprocating stroke begins in the negative direction. **+1**: The starting position is at the cylinder floor. The reciprocating stroke begins in the positive direction. Input: **-1**, **0**, **+1**
	Q1021 One-sided infeed (0/1)? Position at which the lateral infeed occurs: **0**: Lower and upper lateral infeed **1**: One-sided infeed depending on **Q1031** ■ If **Q1031 = -1**, then the lateral infeed is performed above. ■ If **Q1031 = +1**, then the lateral infeed is performed below. Input: **0**, **1**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q1020 Number of idle strokes? Number of idle strokes after the last lateral infeed without material removal. Input: **0...99**
	Q1032 Factor for pitch of helix? The pitch per helical path (= 360°) arises from the factor **Q1032**. **Q1032** is multiplied by the width **B** of the grinding tool. The feed rate for the reciprocating stroke is influenced by the pitch of the helical path. **Further information:** "Feed rate for the reciprocating stroke", Page 945 Input: **0.000...1000**
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q15 Up-cut / climb grinding (-1/+1)? Define the type of contour grinding: **+1**: Climb grinding **-1** or **0**: Up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q211 Idle runs at depth? Number of idle runs at the lower reversal point of the reciprocating stroke. **Further information:** "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 945. Input: **0...99.99**
	Q210 Idle runs at top? Number of idle runs at the upper reversal point of the reciprocating stroke. **Further information:** "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 945. Input: **0...99.99**

Example

```
11 CYCL DEF 1021 CYLINDER, SLOW-STROKE GRINDING ~
    Q650=+0                 ;FIGURE TYPE ~
    Q223=+50                ;FINISHED PART DIA. ~
    Q368=+0.1               ;OVERSIZE AT START ~
    Q14=+0                  ;ALLOWANCE FOR SIDE ~
    Q367=+0                 ;POCKET POSITION ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q1030=+2                ;VERSATZ OBERFLAECHE ~
    Q201=-20                ;DEPTH ~
    Q1031=+1                ;MACHINING DIRECTION ~
    Q1021=+0                ;ONE-SIDED INFEED ~
    Q534=+0.01              ;LATERAL INFEED ~
    Q1020=+0                ;IDLE STROKES ~
    Q1032=+0.5              ;FAKTOR ZUSTELLUNG ~
    Q207=+2000              ;GRINDING FEED RATE ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q15=-1                  ;TYPE OF GRINDING ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q211=+0                 ;IDLE RUNS AT DEPTH ~
    Q210=+0                 ;IDLE RUNS AT TOP
```

15.5.13 Cycle 1022 CYLINDER, FAST-STROKE GRINDING (option 156)

ISO programming
G1022

Application

Refer to your machine manual!
This function must be enabled and adapted by the machine manufacturer.

With the cycle **1022 CYLINDER, FAST STROKE GRINDING**, you can grind circular pockets and circular studs. In the process, the control executes circular and helical paths in order to completely machine the cylinder surface. In order to achieve the required accuracy and surface quality, you can overlay the movement with a reciprocating stroke. The feed rate of the reciprocating stroke is usually so large that multiple reciprocating strokes per circular path are executed. This is equivalent to grinding with a rapid stroke. The lateral infeeds occur above or below depending on the definition. You can program the feed rate of the reciprocating stroke in the cycle.

Cycle sequence

1. The control positions the tool above the cylinder based on the **POCKET POSITION Q367**. At **FMAX**, the control then moves the tool to the **CLEARANCE HEIGHT Q260**.
2. At **FMAX**, the tool moves to the starting point in the working plane and then at **F PRE-POSITIONING Q253** to the **SET-UP CLEARANCE Q200**.
3. The grinding tool moves to the starting point in the tool axis. The starting point depends on the **MACHINING DIRECTION Q1031**. If you have defined a reciprocating stroke in **Q1000**, then the control starts the reciprocating stroke.
4. Depending on the parameter **Q1021**, the control laterally infeeds the grinding tool. The control then infeeds in the tool axis.

   Further information: "Infeed", Page 952
5. If the final depth has been reached, then the grinding tool moves for another full circle without a tool axis infeed.
6. The control repeats steps 4 and 5 until the diameter of the finished part **Q223** or the oversize **Q14** has been reached.
7. After the last infeed run, the grinding tool executes the **IDLE RUNS, CONT. END Q457**.
8. The grinding tool leaves the cylinder on a semi-circular path to the safety clearance **Q200** and stops the reciprocating stroke.
9. At **F PRE-POSITIONING Q253**, the control moves the tool to the **SAFETY CLEARANCE Q200** and then in rapid traverse to the **CLEARANCE HEIGHT Q260**.

Infeed

1 The control infeeds the grinding tool in a semi-circle to the **LATERAL INFEED Q534**.
2 The grinding tool executes a full circle and performs any programmed **IDLE RUNS, CONTOUR Q456**.
3 If the area to be traversed in the tool axis is greater than the grinding wheel width **B**, then the cycle moves in a helical path.

Helical path

You can influence the helical path via a pitch in the parameter **Q1032**. The pitch per helical path (= 360°) is relative to the grinding wheel width.

The number of helical paths (= 360°) depends on the pitch and the **DEPTH Q201**. The smaller the pitch, the more helical paths (= 360°) there are.

Example:

- Grinding wheel width **B** = 20 mm
- **Q201 DEPTH** = 50 mm
- **Q1032 PITCH FACTOR** (pitch) = 0.5

The control calculates the relationship between the pitch relative to the grinding wheel width.

Pitch per helical path = $20mm * 0.5 = 10mm$

The control covers the distance of 10 mm in the tool axis within a helix. The **DEPTH Q201** and the pitch per helical path result in five helical paths.

Number of helical paths = $\frac{50mm}{10mm} = 5$

Notes

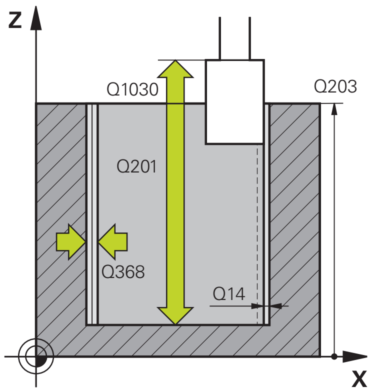
⚙	The overrides for the reciprocation movements can be changed by the machine manufacturer.

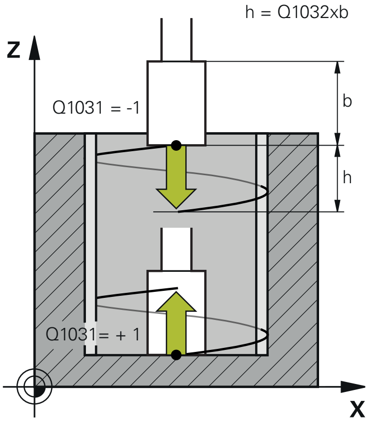
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control always starts the reciprocating stroke in the positive direction.
- The last lateral infeed may be smaller depending on the input.
- The control does not depict the reciprocating movement in the simulation. The reciprocating movement is depicted in the simulation graphics in the **Program run, single block** and **Program run, full sequence** operating modes.
- You can also execute this cycle with a milling cutter. In the case of a milling cutter, the tooth length **LCUTS** equals the width of the grinding wheel.

Notes on programming

- The control assumes that the bottom of the cylinder has a floor. For this reason, you can define an overshoot in **Q1030** only at the surface. If you machine a through hole, for example, then you must take into account the lower overshoot in **DEPTH Q201**.
- If **Q1000=0**, then the control does not execute a superimposed reciprocating movement.

Cycle parameters

Help graphic	Parameter
	Q650 Type of figure? Geometry of the figure: **0**: Pocket **1**: Island Input: **0**, **1**
	Q223 Finished part diameter? Diameter of the fully machined cylinder Input: **0...99999.9999**
	**Q368 Side oversize before machining?** Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q367 Position of pocket (0/1/2/3/4)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool pos. = Center of figure **1**: Tool pos. = Quadrant transition at 90° **2**: Tool pos. = Quadrant transition at 0° **3**: Tool pos. = Quadrant transition at 270° **4**: Tool pos. = Quadrant transition at 180° Input: **0**, **1**, **2**, **3**, **4**
	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**
	Q1030 Offset to surface? Position of the upper edge of the tool on the surface. The offset serves as the overshoot path on the surface for the reciprocating stroke. This value has an absolute effect. Input: **0...999.999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**

Help graphic	Parameter
	**Q1031 Machining direction?** Definition of the machining direction. The starting position arises from this. **-1** or **0**: The control machines the contour from up to down during the first infeed cut. **+1**: The control machines the contour from up to down during the first infeed cut. Input: **-1**, **0**, **+1**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q1032 Factor for pitch of helix? You can define the pitch of the helical path (= 360°) with the factor **Q1032**. This results in the infeed depth per helical path (= 360°). **Q1032** is multiplied by the width **B** of the grinding tool. Input: **0.000...1000**
	Q456 Idle runs around contour? Number of times the grinding tool executes the contour without removing material after every infeed. Input: **0...99**
	Q457 Idle runs at contour end? Number of times the grinding tool executes the contour without material removal after the last infeed. Input: **0...99**
	Q1000 Length of reciprocating stroke? Length of the reciprocating movement, parallel to the active tool axis **0**: The control does not perform a reciprocating motion. Input: **0...9999.9999**
	Q1001 Feed rate for reciprocation? Speed of the reciprocating stroke in mm/min Input: **0...999999**
	Q1021 One-sided infeed (0/1)? Position at which the lateral infeed occurs: **0**: Lower and upper lateral infeed **1**: One-sided infeed depending on **Q1031** ■ If **Q1031 = -1**, then the lateral infeed is performed above. ■ If **Q1031 = +1**, then the lateral infeed is performed below. Input: **0**, **1**

Help graphic	Parameter
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q15 Up-cut / climb grinding (-1/+1)? Define the type of contour grinding: **+1**: Climb grinding **-1** or **0**: Up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 1022 CYLINDER, FAST-STROKE GRINDING ~
    Q650=+0                ;FIGURE TYPE ~
    Q223=+50               ;FINISHED PART DIA. ~
    Q368=+0.1              ;OVERSIZE AT START ~
    Q14=+0                 ;ALLOWANCE FOR SIDE ~
    Q367=+0                ;POCKET POSITION ~
    Q203=+0                ;SURFACE COORDINATE ~
    Q1030=+2               ;SURFACE OFFSET ~
    Q201=-20               ;DEPTH ~
    Q1031=-1               ;MACHINING DIRECTION ~
    Q534=+0.05             ;LATERAL INFEED ~
    Q1032=+0.5             ;PITCH FACTOR ~
    Q456=+0                ;IDLE RUNS, CONTOUR ~
    Q457=+0                ;IDLE RUNS, CONT. END ~
    Q1000=+5               ;RECIPROCATING STROKE ~
    Q1001=+5000            ;RECIP. FEED RATE ~
    Q207=+50               ;GRINDING FEED RATE ~
    Q253=+750              ;F PRE-POSITIONING ~
    Q15=+1                 ;TYPE OF GRINDING ~
    Q260=+100              ;CLEARANCE HEIGHT ~
    Q200=+2                ;SET-UP CLEARANCE
```

15.5.14 Cycle 1025 GRINDING CONTOUR (option 156)

ISO programming
G1025

Application

Use Cycle **1025 GRINDING CONTOUR** in combination with Cycle **14 CONTOUR** to grind open and closed contours.

Cycle sequence

1 The control first moves the tool at rapid traverse to the starting position in the X and Y directions and then to clearance height **Q260**.
2 The tool uses rapid traverse to move to set-up clearance **Q200** above the coordinate surface.
3 From there, it moves at the pre-positioning feed rate **Q253** to the depth **Q201**.
4 If programmed, the control performs the approach movement.
5 The cycle starts with the first stepover **Q534**.
6 If programmed, the control performs the number of idle runs **Q456** after each infeed.
7 This process (steps 5 and 6) is repeated until the contour or finishing allowance **Q14** has been reached.
8 After the last infeed, the specified number of air strokes at contour end **Q457** are performed.
9 The control performs the optional departure movement.
10 Finally, the tool is moved at rapid traverse to the clearance height.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The last stepover may be smaller depending on the input.
- Keep in mind that the cycle takes **M109** or **M110** into account, if programmed. In this case, the control will display the feed rate of the center path of the milling tool. The feed rate shown in the status display may thus become lower for inside radii or become higher for outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 1331

Note on programming

- If you want to program a reciprocating stroke, you need to define and start it before executing this cycle.

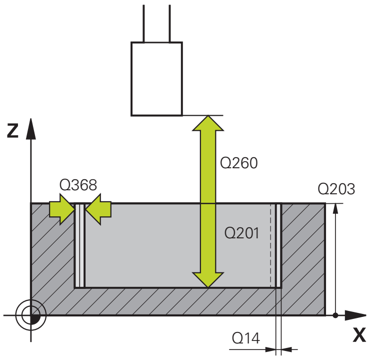
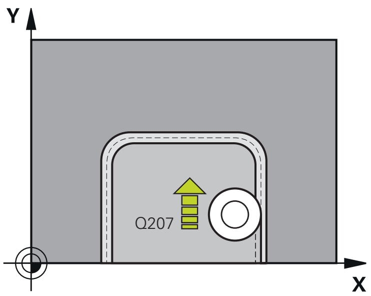
Open contour

- Approach and departure movements for the contour can be programmed using **APPR** and **DEP** or Cycle **270**.

Closed contour

- In the case of a closed contour, only Cycle **270** is available for programming approach and departure movements.
- When grinding a closed contour, it is not possible to alternate between climb and up-cut grinding (**Q15 = 0**). The control issues an error message.
- If you programmed approach and departure movements, the starting position will shift with every infeed. If no approach and departure movements have been programmed, the control automatically generates a vertical movement and the starting position on the contour will not shift.

Cycle parameters

Help graphic	Parameter
	**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**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q368 Side oversize before machining? Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q456 Idle runs around contour? Number of times the grinding tool executes the contour without removing material after every infeed. Input: **0...99**
	Q457 Idle runs at contour end? Number of times the grinding tool executes the contour without material removal after the last infeed. Input: **0...99**
	**Q207 Feed rate for grinding?** Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q15 Up-cut / climb grinding (-1/+1)? Define the machining direction of the contours: **+1**: Climb grinding **-1**: Up-cut grinding **0**: Alternating between climb grinding and up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 1025 GRINDING CONTOUR ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q201=-20                ;DEPTH ~
    Q14=+0                  ;ALLOWANCE FOR SIDE ~
    Q368=+0.1               ;OVERSIZE AT START ~
    Q534=+0.05              ;LATERAL INFEED ~
    Q456=+0                 ;IDLE RUNS, CONTOUR ~
    Q457=+0                 ;IDLE RUNS, CONT. END ~
    Q207=+200               ;GRINDING FEED RATE ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q15=+1                  ;TYPE OF GRINDING ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q200=+2                 ;SET-UP CLEARANCE
```

15.5.15 Cycle 1030 ACTIVATE WHEEL EDGE (option 156)

ISO programming

G1030

Application

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

Use Cycle **1030 ACTIVATE WHEEL EDGE** to activate the desired wheel edge. This means that you can change or update the reference point or reference edge. When dressing, you set the workpiece datum to the corresponding wheel edge with this cycle.

For this cycle, a distinction is made between grinding **(FUNCTION MODE MILL / TURN)** and dressing **(FUNCTION DRESS BEGIN / END)**.

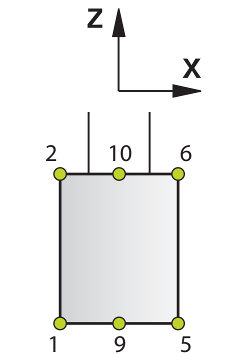
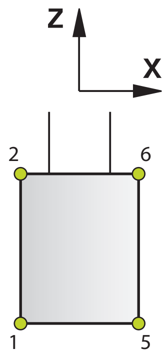
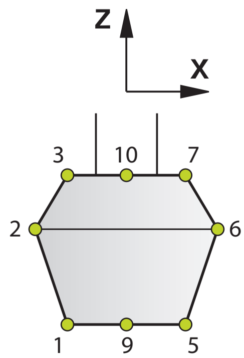
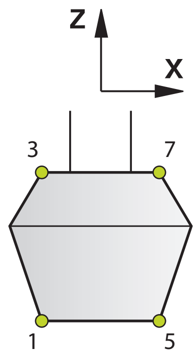
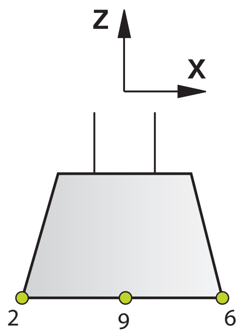
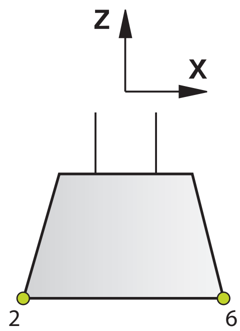
Notes

- This cycle is only permitted in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes if a grinding tool has been activated.
- Cycle **1030** is DEF-active.

Cycle parameters

Help graphic	Parameter
	Q1006 Edge of grinding wheel? Definition of the edge of the grinding tool

Selection of the grinding wheel edges

<table>
<tr><th></th><th>Grinding</th><th>Dressing</th></tr>
<tr><td>Grinding pin</td><td>
</td><td>
</td></tr>
<tr><td>Special grinding pin</td><td>
</td><td>
</td></tr>
<tr><td>Cup wheel</td><td>
</td><td>
</td></tr>
</table>

Example

```
11 CYCL DEF 1030 ACTIVATE WHEEL EDGE ~
    Q1006=+9                ;WHEEL EDGE
```

15.5.16 Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)

ISO programming
G1032

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1032 GRINDING WHL LENGTH COMPENSATION** to define the overall length of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If initial dressing has not been performed (INIT_D_OK = 0), then you can change the basic data. Basic data affect both grinding and dressing.

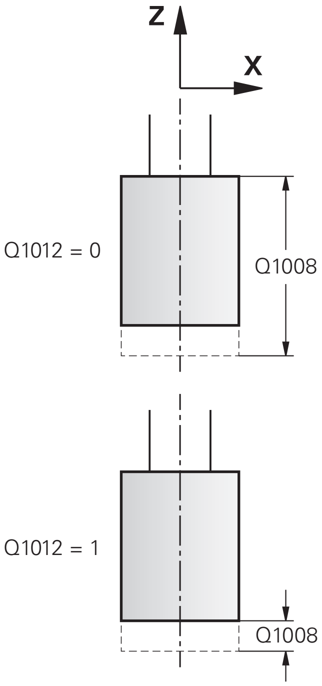
If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

Further information: "Dressing", Page 251

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **1032** is DEF-active.

Cycle parameters

Help graphic	Parameter
	**Q1012 Compens. values (0=abs./1=inc.)?** Definition of the entered length dimension **0**: Entry of the absolute length **1**: Entry of the incremental length Input: **0**, **1**
	Q1008 Comp. value outside edge length? Amount by which the tool is corrected lengthwise based on **Q1012** or by which the tool data are entered without correction. If **Q1012** equals **0**, then the absolute length must be entered. If **Q1012** equals **1**, then the incremental length must be entered. Input: **-999.999...+999.999**
	Q330 Tool number or tool name? Number of name of the grinding tool. Via a selection in the action bar, you have the option of applying the tool directly from the tool table. **-1**: The active tool from the tool spindle is used. Input: **-1...99999.9**

Example

```
11 CYCL DEF 1032 GRINDING WHL LENGTH COMPENSATION ~
   Q1012=+1              ;INCR. COMPENSATION ~
   Q1008=+0              ;COMP. OUTSIDE LENGTH ~
   Q330=-1               ;TOOL
```

15.5.17 Cycle 1033 GRINDING WHL RADIUS COMPENSATION (option 156)

ISO programming
G1033

Application

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

Use Cycle **1033 GRINDING WHL RADIUS COMPENSATION** to define the radius of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If initial dressing has not been performed (INIT_D_OK = 0), then you can change the basic data. Basic data affect both grinding and dressing.

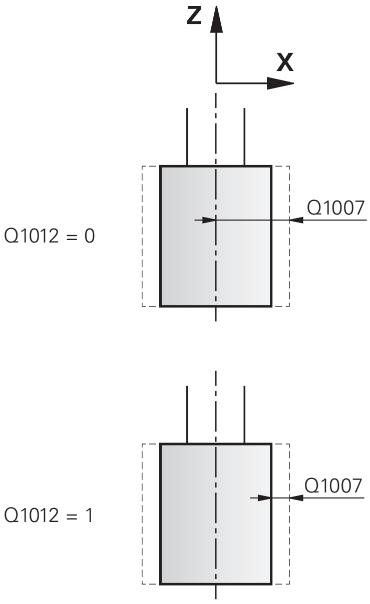
If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

Further information: "Dressing", Page 251

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **1033** is DEF-active.

Cycle parameters

Help graphic	Parameter
	**Q1012 Compens. values (0=abs./1=inc.)?** Definition of the entered radius dimension **0**: Entry of the absolute radius **1**: Entry of the incremental radius Input: **0**, **1**
	Q1007 Compensation value for radius? Dimension by which the tool radius is compensated for based on **Q1012**. If **Q1012** equals **0**, then the absolute radius must be entered. If **Q1012** equals **1**, then the incremental radius must be entered. Input: **-999.9999...+999.9999**
	Q330 Tool number or tool name? Number of name of the grinding tool. Via a selection in the action bar, you have the option of applying the tool directly from the tool table. **-1**: The active tool from the tool spindle is used. Input: **-1...99999.9**

Example

11 CYCL DEF 1033 GRINDING WHL RADIUS COMPENSATION ~	
Q1012=+1	**;INCR. COMPENSATION ~**
Q1007=+0	**;RADIUS COMPENSATION ~**
Q330=-1	**;TOOL**

15.5.18 Programming examples

Example of grinding cycles

This programming example illustrates how to machine with a grinding tool.

The NC program uses the following grinding cycles:

- Cycle **1000 DEFINE RECIP. STROKE**
- Cycle **1002 STOP RECIP. STROKE**
- Cycle **1025 GRINDING CONTOUR**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle **1000 DEFINE RECIP. STROKE**
- Define Cycle **14 CONTOUR**
- Define Cycle **1025 GRINDING CONTOUR**
- Define Cycle **1002 STOP RECIP. STROKE**

0 BEGIN PGM GRINDING_CYCLE MM	
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33	
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1	
3 FUNCTION MODE MILL	
4 TOOL CALL 501 Z S20000	; Tool call: grinding tool
5 L Z+30 R0 FMAX M3	
6 CYCL DEF 1000 DEFINE RECIP. STROKE ~	
Q1000=+13 ;RECIPROCATING STROKE ~	
Q1001=+25000 ;RECIP. FEED RATE ~	
Q1002=+1 ;RECIPROCATION TYPE ~	
Q1004=+1 ;START RECIP. STROKE	
7 CYCL DEF 14.0 CONTOUR	
8 CYCL DEF 14.1 CONTOUR LABEL1 /2	
9 CYCL DEF 14.2	
10 CYCL DEF 1025 GRINDING CONTOUR ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-12 ;DEPTH ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q368=+0.2 ;OVERSIZE AT START ~	
Q534=+0.05 ;LATERAL INFEED ~	
Q456=+2 ;IDLE RUNS, CONTOUR ~	
Q457=+3 ;IDLE RUNS, CONT. END ~	
Q207=+200 ;GRINDING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q15=+1 ;TYPE OF GRINDING ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q200=+2 ;SET-UP CLEARANCE	
11 CYCL CALL	; Cycle call: grinding contour

12 L Z+50 R0 FMAX	
13 CYCL DEF 1002 STOP RECIP. STROKE ~	
Q1005=+1 ;CLEAR RECIP. STROKE ~	
Q1010=+0 ;RECIP.STROKE STOPPOS	
14 L Z+250 R0 FMAX	
15 L C+0 R0 FMAX M92	
16 M30	; End of program
17 LBL 1	; Contour subprogram 1
18 L X+3 Y-23 RL	
19 L X-3	
20 CT X-9 Y-16	
21 CT X-7 Y-10	
22 CT X-7 Y+10	
23 CT X-9 Y+16	
24 CT X-3 Y+23	
25 L X+3	
26 CT X+9 Y+16	
27 CT X+7 Y+10	
28 CT X+7 Y-10	
29 CT X+9 Y-16	
30 CT X+3 Y-23	
31 LBL 0	
32 LBL 2	; Contour subprogram 2
33 L X-25 Y-40 RR	
34 L Y+40	
35 L X+25	
36 L Y-40	
37 L X-25	
38 LBL 0	
39 END PGM GRINDING_CYCLE MM	

Example of dressing cycles

This programming example illustrates dressing mode.

The NC program uses the following grinding cycles:

- Cycle **1030 ACTIVATE WHEEL EDGE**
- Cycle **1010 DRESSING DIAMETER**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle **1030 ACTIVATE WHEEL EDGE**
- Tool call: Dressing tool (no mechanical tool change; only a calculated switch-over)
- Cycle **1010 DRESSING DIAMETER**
- Activate **FUNCTION DRESS END**

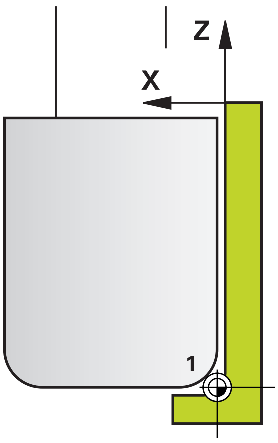
0 BEGIN PGM DRESS_CYCLE MM	
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33	
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1	
3 FUNCTION MODE MILL	
4 TOOL CALL 501 Z S20000	; tool call, grinding wheel
5 M140 MB MAX	
6 L Z+200 R0 FMAX M3	
7 FUNCTION DRESS BEGIN	; activate dressing procedure
8 CYCL DEF 1030 ACTIVATE WHEEL EDGE ~	
Q1006=+5 ;WHEEL EDGE	
9 TOOL CALL 507	; tool call, dressing tool
10 L X+5 R0 F2000	
11 L Y+0 R0	
12 L Z-5 M8	
13 CYCL DEF 1010 DRESSING DIAMETER ~	
Q1013=+0 ;DRESSING AMOUNT ~	
Q1018=+300 ;DRESSING FEED RATE ~	
Q1016=+1 ;DRESSING STRATEGY ~	
Q1019=+2 ;NUMBER INFEEDS ~	
Q1020=+3 ;IDLE STROKES ~	
Q1022=+0 ;COUNTER FOR DRESSING ~	
Q330=-1 ;TOOL ~	
Q1011=+0 ;FACTOR VC	
14 FUNCTION DRESS END	; deactivate dressing procedure
15 M30	; end of program
16 END PGM DRESS_CYCLE MM	

Example of a profile program

Grinding wheel edge no. 1

This example program is for dressing a profile of a grinding wheel. The grinding wheel is curved by the amount of a radius on its outer side.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)



Data to be used:

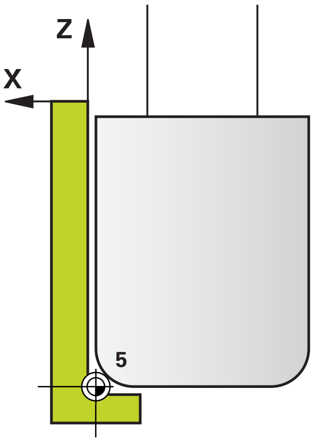
- Grinding wheel edge: 1
- Retraction amount: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm

0 BEGIN PGM 11 MM	
1 L X-5 Z-5 R0 FMAX	; Approach starting position
2 L Z+45 RL FMAX	; Approach starting position
3 L X+0 FQ1018	; Q1018 = Dressing feed rate
4 L Z+0 FQ1018	; Approach radius edge
5 RND R2 FQ1018	; Rounding
6 L X+6 FQ1018	; Approach final position X
7 L Z-5 FQ1018	; Approach final position Z
8 L X-5 Z-5 R0 FMAX	; Approach starting position
9 END PGM 11 MM	

Grinding wheel edge no. 5

This example program is for dressing a profile of a grinding wheel. The grinding wheel is curved by the amount of a radius on its outer side.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)



Data to be used:

- Grinding wheel edge: 5
- Retraction amount: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm

0 BEGIN PGM 12 MM	
1 L X+5 Z-5 R0 FMAX	; Approach starting position
2 L Z+45 RR FMAX	; Approach starting position
3 L X+0 FQ1018	; Q1018 = Dressing feed rate
4 L Z+0 FQ1018	; Approach radius edge
5 RND R2 FQ1018	; Rounding
6 L X-6 FQ1018	; Approach final position X
7 L Z-5 FQ1018	; Approach final position Z
8 L X+5 Z-5 R0 FMAX	; Approach starting position
9 END PGM 11 MM	

15.6 Cycles for gear cutting

15.6.1 Overview

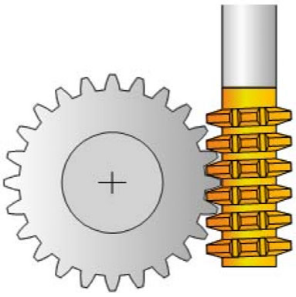
Cycle			Further information
880	**GEAR HOBBING** (option 50 & 131) ■ Description of the geometry and the tool ■ Selection of machining strategy and machining side	**CALL-** active	"Cycle 880 GEAR HOBBING (option 131)"
285	**DEFINE GEAR** (option 157) ■ Define the geometry of the gear wheel	**DEF-** active	"Cycle 285 DEFINE GEAR (option 157)"
286	**GEAR HOBBING** (option 157) ■ Definition of the tool data ■ Selection of the machining strategy and side ■ Possibility of using the entire cutting edge	**CALL-** active	"Cycle 286 GEAR HOBBING (option 157)"
287	**GEAR SKIVING** (option 157) ■ Definition of the tool data ■ Selection of the machining side ■ Definition of the first and last infeed ■ Definition of the number of cuts	**CALL-** active	"Cycle 287 GEAR SKIVING option 157"

15.6.2 Cycle 880 GEAR HOBBING (option 131)

ISO programming
G880

Application

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



With Cycle **880 GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. In the cycle you first define the **gear** and then the **tool** with which the gear is to be machined. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary motion of the tool spindle and rotary table. In addition, the gear hob moves along the workpiece in axial direction.

While Cycle **880 GEAR HOBBING** is active, the coordinate system might be rotated. It is therefore essential to program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145** after the end of the cycle.

Cycle sequence

1 The control positions the tool in the tool axis to clearance height **Q260** at the feed rate FMAX. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the FMAX feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control then tilts the working plane at the feed rate **Q253**; **M144** is internally active in the cycle
4 The control positions the tool at the feed rate FMAX to the starting point in the working plane.
5 The control then moves the tool in the tool axis at the feed rate **Q253** to set-up clearance **Q460**.
6 The control now moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q460** and the end point in Z **Q552**+**Q460**.
7 When the control reaches the end point, it retracts the tool at the feed rate **Q253** and positions it back to the starting point
8 The control repeats the steps 5 to 7 until the defined gear is completed.
9 Finally the control positions the tool to the clearance height **Q260** at the feed rate FMAX
10 The machining operation ends in the tilted system.
11 Now you need to move the tool to a safe height and reset the tilting of the working plane.
12 It is essential that you now program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145**

Notes

NOTICE

Danger of collision!

If you do not position the tool to a safe position, a collision may occur between the tool and workpiece (fixtures) during tilting.

- Pre-position the tool so that it is already on the desired machining side **Q550**.
- Move the tool to a safe position on this machining side

NOTICE

Danger of collision!

If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q460**!

- Clamp the workpiece out of the fixtures far enough to prevent a danger of collision between the tool and the fixtures
- Clamp the workpiece in such a way that its protrusion from the fixture will not cause any collision when the tool is automatically moved to the starting or end point using a path that is extended by the set-up clearance **Q460**

NOTICE

Danger of collision!

Depending on whether you use **M136** or not, the feed rate values will be interpreted differently by the control. If the programmed feed rate was too high, the workpiece might be damaged.

- If you program **M136** explicitly before the cycle, the control will interpret the feed rates in the cycle in mm/rev.
- If you do not program **M136** before the cycle, the control will interpret the feed rates in the cycle in mm/min.

NOTICE

Danger of collision!

If you do not reset the coordinate system after Cycle **880**, the precession angle set by the cycle will remain active. There is a danger of collision!

- Make sure to program Cycle **801** after Cycle **880** in order to reset the coordinate system.
- Make sure to program Cycle **801** after a program abort in order to reset the coordinate system.

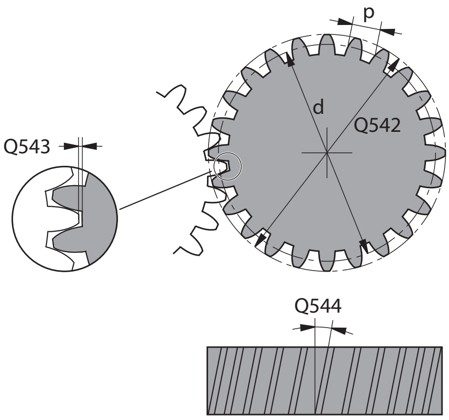
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- Define the tool as a milling cutter in the tool table.
- Before programming the cycle call, set the datum to the center of rotation.

> In order to avoid that the maximum permissible spindle speed of the tool is not exceeded, you can program a limitation. (Specify it in the **Nmax** column of the "tool.t" tool table.)

Notes on programming

- The values entered for the module, number of teeth and outside diameter (outside diameter) are monitored. If these values are not coherent, then an error message is displayed. You can fill in 2 of the 3 parameters. Enter 0 for the module, the number of teeth, or the outside diameter (outside diameter). In this case, the control will calculate the missing value.
- Program FUNCTION TURNDATA SPIN VCONST:OFF.
- If you program FUNCTION TURNDATA SPIN VCONST:OFF S15, then the spindle speed of the tool is calculated as follows: **Q541** x S. With **Q541**=238 and S=15, this would result in a tool spindle speed of 3570 rpm.
- Program the direction of rotation of your workpiece (**M303**/**M304**) before the start of the cycle.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q540 Module? Module of the gear Input: **0...99.999**
	Q541 Number of teeth? Describe gear: number of teeth Input: **0...99999**
	**Q542 Outside diameter?** Describe gear: outside diameter of finished part Input: **0...99999.9999**
	Q543 Trough-to-tip clearance? Distance between the addendum circle of the gear to be made and root circle of the mating gear. Input: **0...9.9999**
	Q544 Angle of inclination? Angle at which the teeth of a helical gear are inclined relative to the direction of the axis. For straight-cut gears, this angle is 0°. Input: **-60...+60**
	Q545 Tool lead angle? Angle of the edges of the gear hob. Enter this value in decimal notation. Example: 0°47'=0.7833 Input: **-60...+60**
	Q546 Reverse tool rotation direction? Describe tool: Direction of spindle rotation of the gear hob **3**: Clockwise rotating tool (**M3**) **4**: Counterclockwise rotating tool (**M4**) Input: **3**, **4**
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**

Help graphic	Parameter
	Q550 Machining side (0=pos./1=neg.)? Define at which side machining is to take place. **0**: Positive machining side of the main axis in the I-CS **1**: Negative machining side of the main axis in the I-CS Input: **0**, **1**
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	Q253 Feed rate for pre-positioning? Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q553 TOOL:L offset, machining start? Define the minimum length offset (L OFFSET) that the tool should have when in use. The control offsets the tool in the longitudinal direction by this amount. This value has an incremental effect. Input: **0...999.999**

Help graphic	Parameter
	Q551 Starting point in Z? Starting point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
	Q552 End point in Z? End point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0,001...999.999**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q488 Feed rate for plunging Feed rate of the tool infeed Input: **0...99999.999** or **FAUTO**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

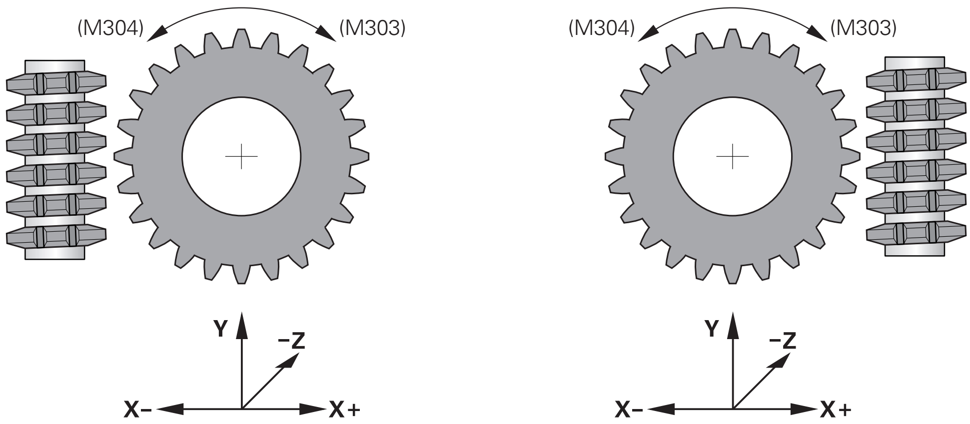
Example

```
11 CYCL DEF 880 GEAR HOBBING ~
    Q215=+0              ;MACHINING OPERATION ~
    Q540=+0              ;MODULE ~
    Q541=+0              ;NUMBER OF TEETH ~
    Q542=+0              ;OUTSIDE DIAMETER ~
    Q543=+0.1666         ;TROUGH-TIP CLEARANCE ~
    Q544=+0              ;ANGLE OF INCLINATION ~
    Q545=+0              ;TOOL LEAD ANGLE ~
    Q546=+3              ;CHANGE TOOL DIRECTN. ~
    Q547=+0              ;ANG. OFFSET, SPINDLE ~
    Q550=+1              ;MACHINING SIDE ~
    Q533=+0              ;PREFERRED DIRECTION ~
    Q530=+2              ;INCLINED MACHINING ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q553=+10             ;TOOL LENGTH OFFSET ~
    Q551=+0              ;STARTING POINT IN Z
    Q552=-10             ;END POINT IN Z
    Q463=+1              ;MAX. CUTTING DEPTH ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q488=+0.3            ;PLUNGING FEED RATE ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q505=+0.2            ;FINISHING FEED RATE
```

Direction of rotation depending on the machining side (Q550)

Determine the direction of rotation of the rotary table:

1. **What tool? (Right-cutting/left-cutting?)**
2. **What machining side? X+ (Q550=0) / X- (Q550=1)**
3. **Look up the direction of rotation of the rotary table in one of the two tables below!** To do so, select the appropriate table for the direction of rotation of your tool (**right-cutting/left-cutting**). Please refer to the tables below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)** ab.



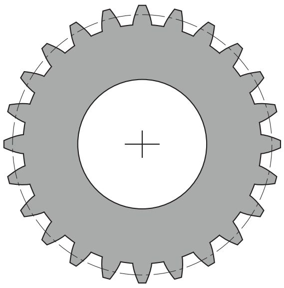
Tool: Right-cutting M3	
Machining side X+ (Q550=0)	Direction of rotation of the table: Clockwise (M303)
Machining side X- (Q550=1)	Direction of rotation of the table: Counterclockwise (M304)

Tool: Left-cutting M4	
Machining side X+ (Q550=0)	Direction of rotation of the table: Counterclockwise (M304)
Machining side X- (Q550=1)	Direction of rotation of the table: Clockwise (M303)

15.6.3 Gear manufacturing fundamentals (option 157)

Fundamentals

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.



For the cycles, option 157 Gear Cutting is required. If you would like to use these cycles in turning mode, you also need option 50. In milling mode, the tool spindle is the master spindle, in turning mode, it is the workpiece spindle. The other spindle is called slave spindle. Depending on the operating mode, you program the speed or the cutting speed with a **TOOL CALL S** or **FUNCTION TURNDATA SPIN**.

To orient the I-CS coordinate system, Cycles **286** and **287** use the precession angle that is also affected by Cycles **800** and **801** in turning mode. At the end of the cycle, the control resets the precession angle to its state at the beginning of the cycle. If one of these cycles is aborted, the precession angle will also be reset.

The axis crossing angle is the angle between workpiece and tool. It results from the angle of inclination of the tool and the angle of inclination of the gear. Based on the required axis crossing angle, Cycles **286** and **287** calculate the required inclination of the rotary axis at the machine. The cycles will always position the first rotary axis starting from the tool.

In order to ensure that the tool can safely be retracted from the gear in the event of a fault (NC stop or power failure), the cycles automatically control the **LiftOff**. The cycles define the direction and path for a **LiftOff**.

The gear itself will first be described in Cycle **285 DEFINE GEAR**. Then, program Cycle **286 GEAR HOBBING** or Cycle **287 GEAR SKIVING**.

Program the following:

- Call a tool with **TOOL CALL**
- Select turning mode or milling mode, with **FUNCTION MODE TURN** or **FUNCTION MODE MILL** "**KINEMATIC_GEAR**" kinematics selection
- Spindle direction of rotation (e.g., **M3** or **M303**)
- Perform pre-positioning for the cycle depending on your selection of **MILL** or **TURN**
- Define the **CYCL DEF 285 DEFINE GEAR** cycle
- Define the **CYCL DEF 286 GEAR HOBBING** or **CYCL DEF 287 GEAR SKIVING** cycle.

Notes

NOTICE
Danger of collision! If you do not pre-position the tool to a safe position, a collision between tool and workpiece (fixtures) may occur during tilting. ▶ Pre-position the tool to a safe position

NOTICE
Danger of collision! If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q200**! ▶ Make sure to clamp the workpiece in such a way that it projects far enough from the fixture and no collision can occur between tool and fixture.

- Before calling the cycle, set the preset to the center of rotation of the workpiece spindle.
- Please note that the slave spindle will continue to rotate after the end of the cycle. If you want to stop the spindle before the end of the program, make sure to program a corresponding M function.
- Activate the **LiftOff** in the tool table. In addition, this function must have been configured by your machine manufacturer.
- Remember that you need to program the speed of the master spindle before calling the cycle, i.e. the tool spindle speed in milling mode and the workpiece spindle speed in turning mode.

Gear formulas

Speed calculation

- n_T: Tool spindle speed
- n_W: Workpiece spindle speed
- z_T: Number of tool teeth
- z_W: Number of workpiece teeth

Definition	Tool spindle	Workpiece spindle
Hobbing	$n_T = n_W * z_W$	$n_W = \frac{n_T}{z_W}$
Skiving	$n_T = n_W * \frac{z_W}{z_T}$	$n_W = n_T * \frac{z_T}{z_W}$

Straight-cut spur gears

- m: Module (**Q540**)
- p: Pitch
- h: Tooth height (**Q563**)
- d: Pitch-circle diameter
- z: Number of teeth (**Q541**)
- c: Trough-to-tip clearance (**Q543**)
- d_a: Diameter of the addendum circle (outside diameter, **Q542**)
- d_f: Root circle diameter

Definition	Formula
Module (**Q540**)	$m = \frac{p}{\pi}$ $m = \frac{d}{z}$
Pitch	$p = \pi * m$
Pitch-circle diameter	$d = m * z$
Tooth height (**Q563**)	$h = 2 * m + c$
Diameter of the addendum circle (outside diameter, **Q542**)	$d_a = m * (z + 2)$ $d_a = d + 2 * m$
Root circle diameter	$d_f = d - 2 * (m + c)$
Root circle diameter if tooth height > 0	$d_f = d_a - 2 * (h + c)$
Number of teeth (**Q541**)	$z = \frac{d}{m}$ $z = \frac{d_a - 2 * m}{m}$

Remember to observe the algebraic sign when calculating an inner gear.

Example: Calculating the diameter of the addendum circle (outside diameter)

Outer gear: **Q540** * (**Q541** + 2) = 1 * (+46 + 2)

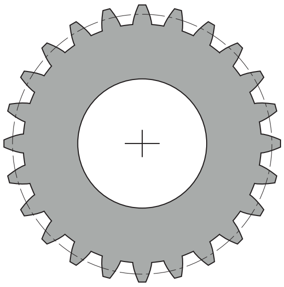
Inner gear: **Q540** * (**Q541** + 2) = 1 * (-46 + 2)

15.6.4 Cycle 285 DEFINE GEAR (option 157)

ISO programming
G285

Application

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



Use Cycle **285 DEFINE GEAR** to describe the geometry of the gearing system. To describe the tool, use Cycle **286 GEAR HOBBING** or Cycle **287GEAR SKIVING** and the tool table (TOOL.T).

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle is DEF-active. The values of these Q parameters will only be read when a CALL-active machining cycle is executed. If you overwrite these input parameters after the cycle definition and before calling the machining cycle, the gear geometry will be modified.
- Define the tool as a milling cutter in the tool table.

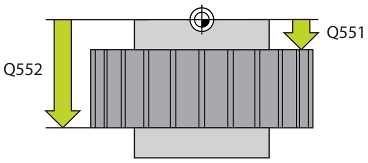
Notes on programming

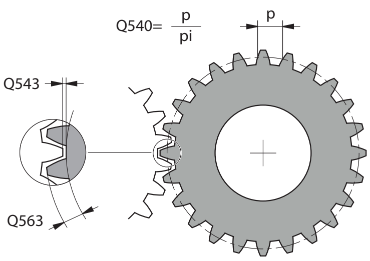
- You must specify values for module and number of teeth. If the outside diameter (diameter of the addendum circle) and the tooth height are defined as 0, normal running gears (DIN 3960) will be machined. If you want to machine gearing systems that differ from this standard, define the corresponding geometry by specifying the diameter of the addendum circle (outside diameter) **Q542** and the tooth height **Q563**.
- If the algebraic signs of the two input parameters **Q541** and **Q542** are contradictory, the cycle will be aborted with an error message.
- Remember that the diameter of the addendum circle is always greater than the root circle diameter, even for an inner gear.

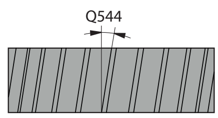
  Inner gear example: The outside diameter (addendum circle) is –40 mm, the root circle diameter is –45 mm. Also in this case, the diameter of the addendum circle (outside diameter) is (numerically) greater than the root circle diameter.

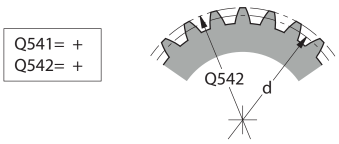
Cycle parameters

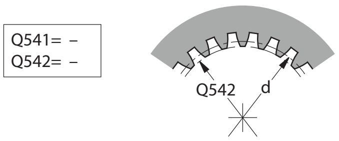
Help graphic











$Q541 = \frac{d}{Q540}$

Q542= Q540x(Q541+2)

Parameter
Q551 Starting point in Z? Starting point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
Q552 End point in Z? End point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
Q540 Module? Module of the gear Input: **0...99.999**
Q541 Number of teeth? Number of teeth. This parameter depends on **Q542**. **+**: If the number of teeth is positive, and at the same time the parameter **Q542** is positive, then an external gear will be machined. **-**: If the number of teeth is negative, and at the same time the parameter **Q542** is negative, then an internal gear will be machined. Input: **-99999...+99999**
Q542 Outside diameter? Addendum circle (outside diameter) of the gear. This parameter depends on **Q541**. **+**: If the addendum circle is positive, and at the same time the parameter **Q541** is positive, then an external gear will be machined. **–**: If the addendum circle is negative, and at the same time the parameter **Q541** is negative, then an internal gear will be machined. Input: **-9999.9999...+9999.9999**
Q563 Tooth height? Distance from the tooth trough to the tooth tip. Input: **0...999.999**
Q543 Trough-to-tip clearance? Distance between the addendum circle of the gear to be made and root circle of the mating gear. Input: **0...9.9999**
Q544 Angle of inclination? Angle at which the teeth of a helical gear are inclined relative to the direction of the axis. For straight-cut gears, this angle is 0°. Input: **-60...+60**

Example

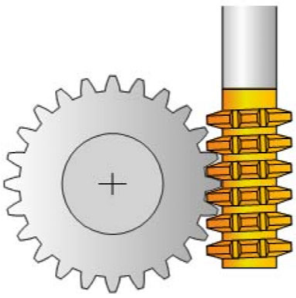
11 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0	;STARTING POINT IN Z ~
Q552=-10	;END POINT IN Z ~
Q540=+1	;MODULE ~
Q541=+10	;NUMBER OF TEETH ~
Q542=+0	;OUTSIDE DIAMETER ~
Q563=+0	;TOOTH HEIGHT ~
Q543=+0.17	;TROUGH-TIP CLEARANCE ~
Q544=+0	;ANGLE OF INCLINATION

15.6.5 Cycle 286 GEAR HOBBING (option 157)

ISO programming
G286

Application

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



With Cycle **286 GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction. Both for roughing and for finishing, the cutting operation may be offset by x edges relative to a height defined at the tool (e.g., 10 cutting edges for a height of 10 mm). This means that all cutting edges will be used in order to increase the tool life of the tool.

Cycle sequence

1 The control positions the tool in the tool axis to clearance height **Q260** at the feed rate **FMAX**. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the **FMAX** feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control then tilts the working plane at the feed rate **Q253**
4 The control positions the tool at the feed rate **FMAX** to the starting point in the working plane
5 The control then moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**.
6 The control moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552**+**Q200** (**Q551** and **Q552** are defined in Cycle **285**).
   Further information: "Cycle 285 DEFINE GEAR (option 157)", Page 984
7 When the tool reaches the end point, it is retracted at the feed rate **Q253** and returns to the starting point.
8 The control repeats the steps 5 to 7 until the defined gear is completed.
9 Finally, the control retracts the tool to the clearance height **Q260** at the feed rate **FMAX**.

Notes

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

▶ Make sure to retract the tool before changing the position of the tilting axis

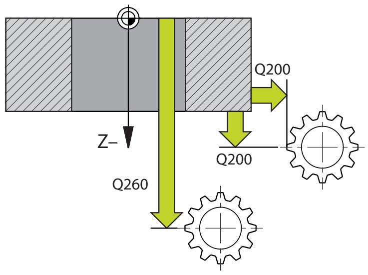
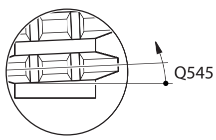
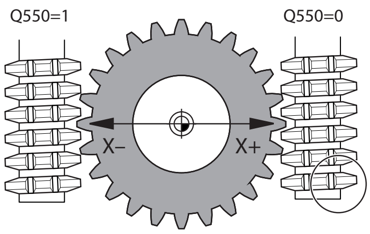
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- The maximum speed of the rotary table cannot be exceeded. If you have specified a higher value under **NMAX** in the tool table, the control will decrease the value to the maximum speed.

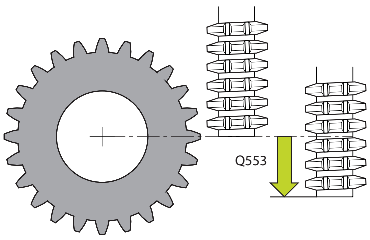
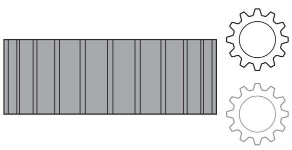
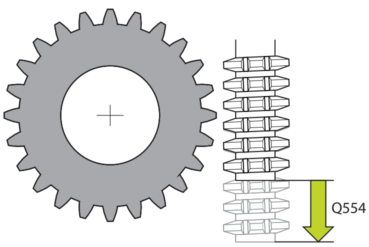
> Avoid master spindle speeds of less than 6 rpm. Otherwise, it is not possible to reliably use a feed rate in mm/rev.

Notes on programming

- In order to ensure constant engagement of the cutting edge of a tool, you need to define a very small path in cycle parameter **Q554 SYNCHRONOUS SHIFT**.
- Make sure to program the direction of rotation of the master spindle (channel spindle) before the cycle start.
- If you program **FUNCTION TURNDATA SPIN VCONST:OFF S15**, the spindle speed of the tool is calculated as **Q541** x S. With **Q541** = 238 and S = 15, this would result in a tool spindle speed of 3570 rpm.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q200 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
 	**Q545 Tool lead angle?** Angle of the edges of the gear hob. Enter this value in decimal notation. Example: 0°47'=0.7833 Input: **-60...+60**
	Q546 Reverse spindle rotation dir.? Direction of rotation of the slave spindle: **0**: No change in the direction of rotation **1**: Change in the direction of rotation Input: **0**, **1** **Further information:** "Verifying and changing directions of rotation of the spindles", Page 992
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**
	Q550 Machining side (0=pos./1=neg.)? Define at which side machining is to take place. **0**: Positive machining side of the main axis in the I-CS **1**: Negative machining side of the main axis in the I-CS Input: **0**, **1**

Help graphic	Parameter
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	Q253 Feed rate for pre-positioning? Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	**Q553 TOOL:L offset, machining start?** Define the minimum length offset (L OFFSET) that the tool should have when in use. The control offsets the tool in the longitudinal direction by this amount. This value has an incremental effect. Input: **0...999.999**
	**Q554 Path for synchronous shift?** Define by which distance the gear hob will be offset in its axial direction during machining. This way, tool wear can be distributed over this area of the cutting edges. For helical gears, it is thus possible to limit the cutting edges used for machining. Entering **0** deactivates the synchronous shift function. Input: **-99...+99.9999**

Help graphic	Parameter
	Q548 Tool shift for roughing? Specify the number of cutting edges by which the control will shift the roughing tool in its axial direction. The shift will be performed incrementally relative to parameter **Q553**. Entering 0 deactivates the shift function. Input: **-99...+99**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0,001...999.999**
	Q488 Feed rate for plunging Feed rate for tool infeed. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q478 Roughing feed rate? Feed rate during roughing. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q549 Tool shift for finishing? Specify the number of cutting edges by which the control will shift the finishing tool in its longitudinal direction. The shift will be performed incrementally relative to parameter **Q553**. Entering 0 deactivates the shift function. Input: **-99...+99**

Example

```
11 CYCL DEF 286 GEAR HOBBING ~
    Q215=+0          ;MACHINING OPERATION ~
    Q200=+2          ;SET-UP CLEARANCE ~
    Q260=+100        ;CLEARANCE HEIGHT ~
    Q545=+0          ;TOOL LEAD ANGLE ~
    Q546=+0          ;CHANGE ROTATION DIR. ~
    Q547=+0          ;ANG. OFFSET, SPINDLE ~
    Q550=+1          ;MACHINING SIDE ~
    Q533=+0          ;PREFERRED DIRECTION ~
    Q530=+2          ;INCLINED MACHINING ~
    Q253=+750        ;F PRE-POSITIONING ~
    Q553=+10         ;TOOL LENGTH OFFSET ~
    Q554=+0          ;SYNCHRONOUS SHIFT ~
    Q548=+0          ;ROUGHING SHIFT ~
    Q463=+1          ;MAX. CUTTING DEPTH ~
    Q488=+0.3        ;PLUNGING FEED RATE ~
    Q478=+0.3        ;ROUGHING FEED RATE ~
    Q483=+0.4        ;OVERSIZE FOR DIAMETER ~
    Q505=+0.2        ;FINISHING FEED RATE ~
    Q549=+0          ;FINISHING SHIFT
```

Verifying and changing directions of rotation of the spindles

Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Determine the direction of rotation of the rotary table:

1 What tool? (Right-cutting/left-cutting?)
2 Which machining side? **X+ (Q550=0) / X- (Q550=1)**
3 Look up the direction of rotation of the rotary table in one of the two tables below! To do so, select the appropriate table for the direction of rotation of your tool (right-cutting/left-cutting). Please refer to the appropriate table below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)**.

Tool: Right-cutting M3

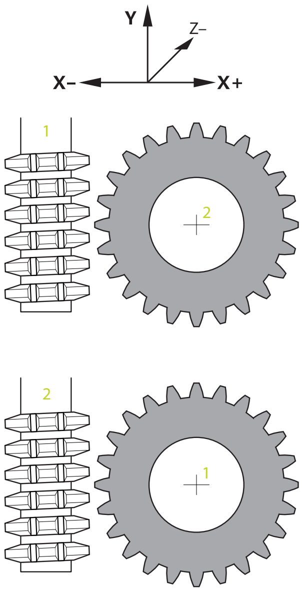
Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Clockwise (e.g., **M303**)
X- (Q550=1)	Counterclockwise (e.g., **M304**)

Tool: Left-cutting M4

Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Counterclockwise (e.g., **M304**)
X- (Q550=1)	Clockwise (e.g., **M303**)

Keep in mind that in special cases, the directions of rotation might deviate from the ones indicated in these tables.

Changing the direction of rotation



Milling:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Turning:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

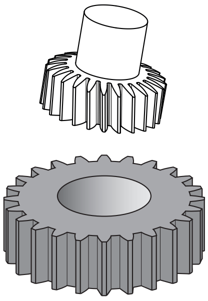
> Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.
>
> If required, define a low spindle speed to make sure that the direction of rotation is correct.

15.6.6 Cycle 287 GEAR SKIVING option 157

ISO programming
G287

Application

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



With Cycle **287 GEAR SKIVING**, you can machine cylindrical gears or helical gears with any angles. Cutting takes place on the one hand by the axial feeding of the tool and on the other hand through the rolling motion.

You can select the machining side in the cycle. The machining process for gear skiving is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction.

In the cycle, you can call a table containing technology data. In this table, you can define the feed rate, the lateral infeed, and the lateral offset for each cut.

Further information: "Technology table for Cycle 287 Gear Skiving", Page 2066

Cycle sequence

1 The control positions the tool in the tool axis to the clearance height **Q260** at the feed rate **FMAX**. If the value of the current tool position in the tool axis is greater than **Q260**, the tool will not be moved
2 Before tilting the working plane, the control positions the tool in X at the feed rate **FMAX** to a safe coordinate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control tilts the working plane at the feed rate **Q253**
4 The control positions the tool at the feed rate **FMAX** to the starting point in the working plane
5 After that, the control positions the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**
6 The control then traverses the approach length. The control automatically calculates this distance. The approach length is the distance from the initial scratch to the complete plunging depth.
7 The control rolls the tool over the workpiece to be geared in longitudinal direction at the defined feed rate. During the initial infeed of the cut **Q586** the control moves at the initial feed rate **Q588**. The control then uses intermediate values for the infeed and feed rate of the next cuts. The control calculates these values itself. The intermediate feed rate values, however, depend on the factor for feed-rate adaptation **Q580**. When the control arrives at the last infeed **Q587**, it performs the last cut with the feed rate **Q589**
8 The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552** (**Q551** and **Q552** are defined in Cycle **285**). The approach length must be added to the starting point. Its purpose is to prevent the tool from plunging into the workpiece all the way to the machining diameter. The control calculates this distance itself.
9 At the end of machining, the tool moves beyond the defined end point by the overrun path **Q580**. The overrun path serves to completely machine the gear.
10 When the control reaches the end point, it retracts the tool at the feed rate **Q253** and positions it back to the starting point
11 Finally the control positions the tool to the clearance height **Q260** at the feed rate FMAX

Notes

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

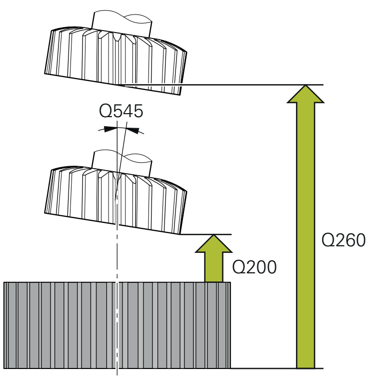
- Make sure to retract the tool before changing the position of the tilting axis

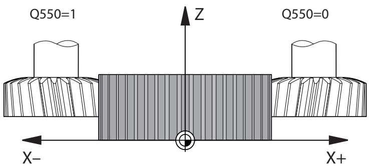
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- The speed ratio between tool and workpiece results from the number of teeth of the gear wheel and the number of cutting edges of the tool.

Notes on programming

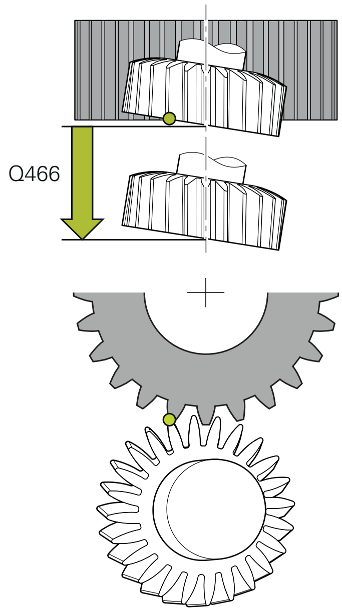
- Make sure to program the direction of rotation of the master spindle (channel spindle) before the cycle start.
- The larger the factor in **Q580 FEED-RATE ADAPTION**, the earlier the control will adapt the feed rate to the feed rate for the last cut. The recommended value is 0.2.
- When defining the tool, make sure to specify the number of cutting edges as indicated in the tool table.
- If only two cuts have been programmed in **Q240**, the last infeed from **Q587** and the last feed rate from **Q589** will be ignored. If only one cut has been programmed, the first infeed from **Q586** will also be ignored.

Cycle parameters

Help graphic	Parameter
	Q240 Number of cuts? Number of cuts to the final depth **0**: The control automatically determines the minimum number of cuts **1**: One cut **2**: Two cuts where the control considers only the infeed for the first cut **Q586**. The control does not consider the infeed for the last cut **Q587**. **3 to 99**: Programmed number of cuts **"..."**: Path of a table containing technology data see "Technology table for Cycle 287 Gear Skiving", Page 2066 Input: **0...99** or text entry of max. **255** characters or **QS** parameter
	Q584 Number of the first cut? Define which cut number the control will perform first. Input: **1...999**
	Q585 Number of the last cut? Define at which number the control will perform the last cut. Input: **1...999**
	**Q200 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q545 Tool lead angle? Angle of the edges of the skiving tool. Enter this value in decimal notation. Example: 0°47' = 0.7833 Input: **-60...+60**

Help graphic	Parameter
	Q546 Reverse spindle rotation dir.? Direction of rotation of the slave spindle: **0**: No change in the direction of rotation **1**: Change in the direction of rotation Input: **0**, **1** **Further information:** "Verifying and changing directions of rotation of the spindles", Page 1000
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**
 	**Q550 Machining side (0=pos./1=neg.)?** Define at which side machining is to take place. **0**: Positive machining side of the main axis in the I-CS **1**: Negative machining side of the main axis in the I-CS Input: **0**, **1**
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	Q253 Feed rate for pre-positioning? Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q586 Infeed for first cut? Infeed for the first cut. This value has an incremental effect. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 2066 Input: **0,001...99.999**
	Q587 Infeed for last cut? Infeed for the last cut. This value has an incremental effect. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 2066 Input: **0,001...99.999**
	Q588 Feed rate for first cut? Feed rate for the first cut. The control interprets the feed rate in mm per workpiece revolution. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 2066 Input: **0,001...99.999**
	Q589 Feed rate for last cut? Feed rate for the last cut. The control interprets the feed rate in mm per workpiece revolution. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 2066 Input: **0,001...99.999**
	Q580 Factor for feed-rate adaptation? Using this factor, you can define a feed rate reduction. This is due to the fact that the feed rate must decrease with increasing cutting numbers. The greater the value, the earlier the control will adapt the feed rates to match the last feed rate. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 2066 Input: **0...1**

Help graphic	Parameter
	**Q466 Overrun path?** Length of overtravel at the end of the gear teeth. The overtravel path ensures that the control machines the gear teeth up to the desired end point. If you do not program these optional parameters, then the control uses the safety clearance **Q200** as the overtravel path. Input: **0.1...99.9**

Example

11 CYCL DEF 287 GEAR SKIVING ~	
Q240=+0	;NUMBER OF CUTS ~
Q584=+1	;NO. OF FIRST CUT ~
Q585=+999	;NO. OF LAST CUT ~
Q200=+2	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q545=+0	;TOOL LEAD ANGLE ~
Q546=+0	;CHANGE ROTATION DIR. ~
Q547=+0	;ANG. OFFSET, SPINDLE ~
Q550=+1	;MACHINING SIDE ~
Q533=+0	;PREFERRED DIRECTION ~
Q530=+2	;INCLINED MACHINING ~
Q253=+750	;F PRE-POSITIONING ~
Q586=+1	;FIRST INFEED ~
Q587=+0.1	;LAST INFEED ~
Q588=+0.2	;FIRST FEED RATE ~
Q589=+0.05	;LAST FEED RATE ~
Q580=+0.2	;FEED-RATE ADAPTION ~
Q466=+2	;OVERRUN PATH

Verifying and changing directions of rotation of the spindles

Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Determine the direction of rotation of the rotary table:

1. What tool? (Right-cutting/left-cutting?)
2. Which machining side? **X+ (Q550=0) / X- (Q550=1)**
3. Look up the direction of rotation of the rotary table in one of the two tables below! To do so, select the appropriate table for the direction of rotation of your tool (right-cutting/left-cutting). Please refer to the appropriate table below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)**.

Tool: Right-cutting M3

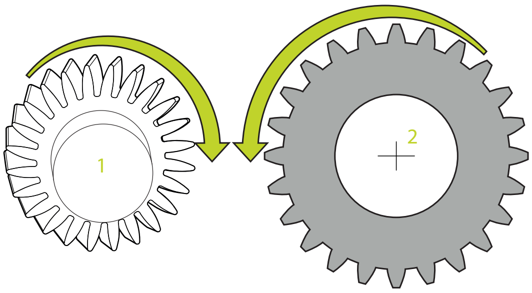
Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Clockwise (e.g., **M303**)
X- (Q550=1)	Counterclockwise (e.g., **M304**)

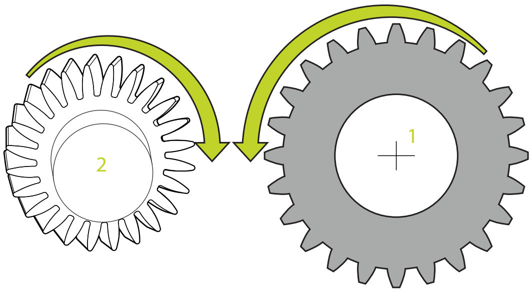
Tool: Left-cutting M4

Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Counterclockwise (e.g., **M304**)
X- (Q550=1)	Clockwise (e.g., **M303**)

Keep in mind that in special cases, the directions of rotation might deviate from the ones indicated in these tables.

Changing the direction of rotation





Milling:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Turning:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

> Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.
>
> If required, define a low spindle speed to make sure that the direction of rotation is correct.

15.6.7 Programming examples

Example: Gear hobbing

The following NC program uses Cycle **880 GEAR HOBBING** This programming example illustrates the machining of a helical gear, with Module=2.1.

Program sequence

- Tool call: Gear hob
- Start turning mode
- Move to safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGIN PGM 8 MM	
1 BLK FORM CYLINDER Z R42 L150	
2 FUNCTION MODE MILL	; Activate milling mode
3 TOOL CALL "GEAD_HOB"	; Call tool
4 FUNCTION MODE TURN	; Activate turning mode
5 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
6 M145	; Cancel a potentially still active M144
7 FUNCTION TURNDATA SPIN VCONST:OFF S50	; Constant cutting speed OFF
8 M140 MB MAX	; Retract the tool
9 L A+0 R0 FMAX	; Set turning axis to 0
10 L X+250 Y-250 R0 FMAX M303	; Pre-position the tool in the working plane on the side on which machining will be performed, Spindle ON
11 L Z+20 R0 FMAX	; Pre-position the tool in the spindle axis
12 M136	; Feed rate in mm/rev.
13 CYCL DEF 880 GEAR HOBBING ~	
Q215=+0 ;MACHINING OPERATION ~	
Q540=+2.1 ;MODULE ~	
Q541=+0 ;NUMBER OF TEETH ~	
Q542=+69.3 ;OUTSIDE DIAMETER ~	
Q543=+0.1666 ;TROUGH-TIP CLEARANCE ~	
Q544=-5 ;ANGLE OF INCLINATION ~	
Q545=+1.6833 ;TOOL LEAD ANGLE ~	
Q546=+3 ;CHANGE TOOL DIRECTN. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+0 ;MACHINING SIDE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q530=+2 ;INCLINED MACHINING ~	
Q253=+800 ;F PRE-POSITIONING ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q553=+10 ;TOOL LENGTH OFFSET ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-10 ;END POINT IN Z ~	

Q463=+1 ;MAX. CUTTING DEPTH ~	
Q460=2 ;SAFETY CLEARANCE ~	
Q488=+1 ;PLUNGING FEED RATE ~	
Q478=+2 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q505=+1 ;FINISHING FEED RATE	
14 CYCL CALL	; Call cycle
15 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
16 M145	; Switch off active M144 in the cycle
17 FUNCTION MODE MILL	; Activate milling mode
18 M140 MB MAX	; Retract tool in the tool axis
19 L A+0 C+0 R0 FMAX	; Reset turning
20 M30	; End of program
21 END PGM 8 MM	

Example of hob milling

The following NC program uses Cycle **286 GEAR HOBBING**. This programming example shows how to machine an involute spline with module = 1 (deviating from DIN 3960).

Program sequence

- Tool call: Gear hob
- Start the turning mode
- Reset the coordinate system with Cycle **801**
- Move to safe position
- Define Cycle **285**
- Call Cycle **286**
- Reset the coordinate system with Cycle **801**

0 BEGIN PGM 7 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI58	
2 TOOL CALL "GEAR_HOB"	; Call the tool
3 FUNCTION MODE TURN	; Activate turning mode
* - ...	; Reset the coordinate system
4 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
5 M145	; Cancel a potentially still active M144
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	; Constant surface speed OFF
7 M140 MB MAX	; Retract the tool
8 L A+0 R0 FMAX	; Set the rotary axis to 0
9 L X+0 Y+0 R0 FMAX	; Pre-position the tool at the workpiece center
10 L Z+50 R0 FMAX	; Pre-position the tool in the spindle axis
11 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-11 ;END POINT IN Z ~	
Q540=+1 ;MODULE ~	
Q541=+90 ;NUMBER OF TEETH ~	
Q542=+90 ;OUTSIDE DIAMETER ~	
Q563=+1 ;TOOTH HEIGHT ~	
Q543=+0.05 ;TROUGH-TIP CLEARANCE ~	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 286 GEAR HOBBING ~	
Q215=+0 ;MACHINING OPERATION ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q260=+30 ;CLEARANCE HEIGHT ~	
Q545=+1.6 ;TOOL LEAD ANGLE ~	
Q546=+0 ;CHANGE ROTATION DIR. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+1 ;MACHINING SIDE ~	
Q533=+1 ;PREFERRED DIRECTION ~	
Q530=+2 ;INCLINED MACHINING ~	

Q253=+2222 ;F PRE-POSITIONING ~	
Q553=+5 ;TOOL LENGTH OFFSET ~	
Q554=+10 ;SYNCHRONOUS SHIFT ~	
Q548=+1 ;ROUGHING SHIFT ~	
Q463=+1 ;MAX. CUTTING DEPTH ~	
Q488=+0.3 ;PLUNGING FEED RATE ~	
Q478=+0.3 ;PLUNGING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q549=+3 ;FINISHING SHIFT	
13 CYCL CALL M303	; Call the cycle, spindle ON
14 FUNCTION MODE MILL	; Activate milling mode
15 M140 MB MAX	; Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	; Reset the rotation
17 M30	; End of program
18 END PGM 7 MM	

Example of skiving

The following NC program uses Cycle **287 GEAR SKIVING**. This programming example shows how to machine an involute spline with module = 1 (deviating from DIN 3960).

Program sequence

- Tool call: Internal gear cutter
- Start turning mode
- Reset the coordinate system with Cycle **801**
- Move to safe position
- Define Cycle **285**
- Call Cycle **287**
- Reset the coordinate system with Cycle **801**

0 BEGIN PGM 7 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI58	
2 TOOL CALL "SKIVING"	; Call the tool
3 FUNCTION MODE TURN	; Activate turning mode
4 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
5 M145	; Cancel a potentially still active M144
6 FUNCTION TURNDATA SPIN VCONST: OFF S50	; Constant surface speed OFF
7 M140 MB MAX	; Retract the tool
8 L A+0 R0 FMAX	; Set the rotary axis to 0
9 L X+0 Y+0 R0 FMAX	; Pre-position the tool at the workpiece center
10 L Z+50 R0 FMAX	; Pre-position the tool in the spindle axis
11 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-11 ;END POINT IN Z ~	
Q540=+1 ;MODULE ~	
Q541=+90 ;NUMBER OF TEETH ~	
Q542=+90 ;OUTSIDE DIAMETER ~	
Q563=+1 ;TOOTH HEIGHT ~	
Q543=+0.05 ;TROUGH-TIP CLEARANCE ~	
Q544=+10 ;ANGLE OF INCLINATION	
12 CYCL DEF 287 GEAR SKIVING ~	
Q240=+5 ;CUTS/TABLE ~	
Q584=+1 ;NO. OF FIRST CUT ~	
Q585=+5 ;NO. OF LAST CUT ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q260=+50 ;CLEARANCE HEIGHT ~	
Q545=+20 ;TOOL LEAD ANGLE ~	
Q546=+0 ;CHANGE ROTATION DIR. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+1 ;MACHINING SIDE ~	
Q533=+1 ;PREFERRED DIRECTION ~	

Q530=+2 ;INCLINED MACHINING ~	
Q253=+2222 ;F PRE-POSITIONING ~	
Q586=+0.4 ;FIRST INFEED ~	
Q587=+0.1 ;LAST INFEED ~	
Q588=+0.4 ;FIRST FEED RATE ~	
Q589=+0.25 ;LAST FEED RATE ~	
Q580=+0.2 ;FEED-RATE ADAPTION ~	
Q466=+2 ;OVERRUN PATH	
13 CYCL CALL M303	; Call the cycle, spindle ON
14 FUNCTION MODE MILL	; Activate milling mode
15 M140 MB MAX	; Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	; Reset the rotation
17 M30	; End of program
18 END PGM 7 MM	

16

Coordinate Transformation

16.1 Reference systems

16.1.1 Overview

A control requires unambiguous coordinates in order to move an axis to a defined position correctly. For coordinates to be unambiguous, they not only require the values but also a reference system in which these values are valid.

The control differentiates between the following reference systems:

Abbreviation	Meaning	Further information
M-CS	Machine coordinate system machine coordinate system	Page 1012
B-CS	Basic coordinate system basic coordinate system	Page 1014
W-CS	Workpiece coordinate system workpiece coordinate system	Page 1016
WPL-CS	Working plane coordinate system working plane coordinate system	Page 1018
I-CS	Input coordinate system input coordinate system	Page 1021
T-CS	Tool coordinate system tool coordinate system	Page 1022

The control uses different reference systems for different purposes. For example, this makes it possible to always exchange tools at the exact same position while maintaining the possibility of adapting an NC program to the workpiece position.

The reference systems build upon each other. The machine coordinate system **M-CS** is the fundamental reference system. The position and orientation of the following reference systems are determined by transformations of the M-CS.

Definition

Transformations

Translatory transformations each enable a shift along a number line. Rotatory transformations enable a rotation around a point.

16.1.2 Basics of coordinate systems

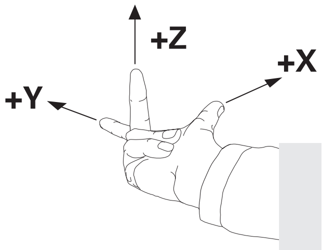
Types of coordinate systems

For coordinates to be unambiguous they must define one point in all axes of the coordinate system:

Axes	Function
One	In a one-dimensional coordinate system, one coordinate defines one point on a number line. Example: on a machine tool, a linear encoder represents a number line.
Two	In a two-dimensional coordinate system, two coordinates define one point in a plane.
Three	In a three-dimensional coordinate system, three coordinates define one point in space.

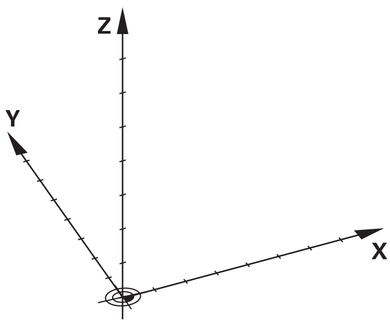
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

Using the right-hand rule you can recreate a three-dimensional Cartesian coordinate system. The fingertips point in the positive directions of the three axes.



Origin of the coordinate system

Unambiguous coordinates require a defined reference point to which the values refer, starting from zero. This point is the coordinate origin, which lies at the intersection of the axes for all three-dimensional Cartesian coordinate systems of the control. The coordinate origin has the coordinates **X+0**, **Y+0**, and **Z+0**.



16.1.3 Machine coordinate system M-CS

Application

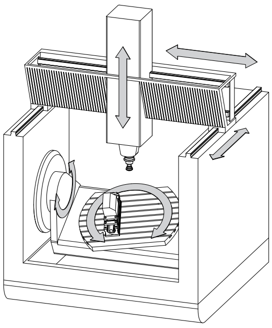
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

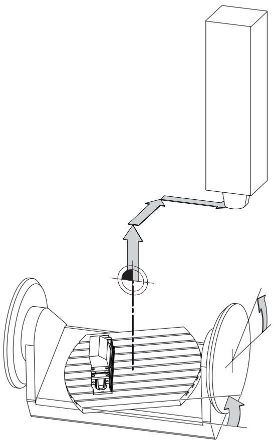
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate system. The **M-CS** thus consists of multiple one-dimensional coordinate systems that correspond to the axes of the machine.

The machine manufacturer defines the position and orientation of the one-dimensional coordinate systems in the kinematics description.



The machine datum is the coordinate origin of the **M-CS**. The machine manufacturer defines the machine datum in the machine configuration.

The values in the machine configuration define the zero positions of the position encoders and the corresponding machine axes. The machine datum does not necessarily have to be located in the theoretical intersection of the physical axes. It can also be located outside of the traverse range.



Position of the machine datum in the machine

Transformations in the machine coordinate system M-CS

The following transformations can be defined in the **M-CS** machine coordinate system:

- Axis-specific shifts in the **OFFS** columns of the preset table

  Further information: "Preset table", Page 2035

  > The machine manufacturer configures the **OFFS** columns of the preset table in accordance with the machine.

- The **Additive offset (M-CS)** function for rotary axes in the **GS** workspace (option 44)

  Further information: "Global Program Settings (GPS, option 44)", Page 1217

> The machine manufacturer can also define further transformations.
>
> **Further information:** "Note", Page 1013

Position display

The following modes of the position display are referenced to the machine coordinate system **M-CS**:

- **Nominal reference position (RFNOML)**
- **Actual reference position (RFACTL)**

The difference between the values for the **RFACTL** and **ACTL.** modes of an axis result from all stated offsets as well as all active transformations in other reference systems.

Programming coordinate entry in machine coordinate system M-CS

With miscellaneous function **M91** you program the coordinates relative to the machine datum.

Further information: "Traversing in the machine coordinate system M-CS with M91", Page 1322

Note

The machine manufacturer can define the following further transformations in the machine coordinate system **M-CS**:

- Additive axis shifts for parallel axes with the **OEM-offset**
- Axis-specific shifts in the **OFFS** columns of the pallet preset table

  Further information: "Pallet preset table", Page 1951

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets

Example

This example illustrates the difference between traverse movements with and without **M91**. The example shows the behavior with a Y axis as oblique axis that is not arranged perpendicularly to the ZX plane.

Traverse movement without M91

```
11 L IY+10
```

You use the Cartesian input coordinate system **I-CS** for programming. The **ACTL.** and **NOML.** modes of the position display show only a movement of the Y axis in the **I-CS**.

The control uses the defined values to determine the required traverse paths of the machine axes. Since the machine axes are not arranged perpendicularly to each other, the control moves the axes **Y** and **Z**.

Since the machine coordinate system **M-CS** is a projection of the machine axes, the **RFACTL** and **RFNOML** modes of the position display show movements of the Y axis and Z axis in the **M-CS**.

Traverse movement with M91

```
11 L IY+10 M91
```

The control moves the machine axis **Y** by 10 mm. The **RFACTL** and **RFNOML** modes of the position display show only a movement of the Y axis in the **M-CS**.

In contrast to the **M-CS**, the **I-CS** is a Cartesian coordinate system; the axes of the two reference systems do not coincide. The **ACTL.** and **NOML.** modes of the position display show movements of the Y axis and Z axis in the **I-CS**.

16.1.4 Basic coordinate system B-CS

Application

In the basic coordinate system **B-CS** you define the position and orientation of the workpiece. You determine these values by using a 3D touch probe, for example. The control saves the values in the preset table.

Description of function

Properties of the basic coordinate system B-CS

The basic coordinate system **B-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the end of the kinematics description.

The machine manufacturer defines the coordinate origin and orientation of the **B-CS**.

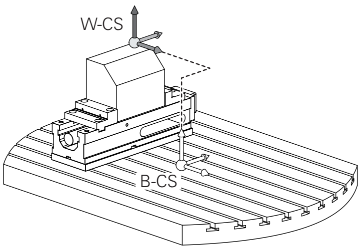
Transformations in the basic coordinate system B-CS

The following columns of the preset table have an effect in the basic coordinate system **B-CS**:

- **X**
- **Y**
- **Z**
- **SPA**
- **SPB**
- **SPC**

You determine the position and orientation of the workpiece coordinate system **W-CS** by using a 3D touch probe, for example. The control saves the determined values as basic transformations in the **B-CS** in the preset table.

Further information: "Preset management", Page 1025



The machine manufacturer configures the **BASE TRANSFORM.** columns of the preset table in accordance with the machine.

Further information: "Note", Page 1015

Note

The machine manufacturer can define additional basic transformations and store them in the pallet preset table.

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets

16.1.5 Workpiece coordinate system W-CS

Application

In the workpiece coordinate system **W-CS** you define the position and orientation of the working plane. You do this by programming transformations and tilting the working plane.

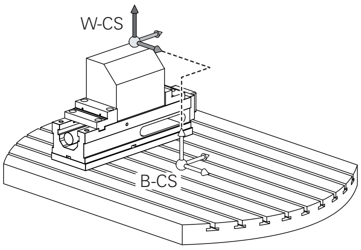
Description of function

Properties of the workpiece coordinate system W-CS

The workpiece coordinate system **W-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the active workpiece preset from the preset table.

Both the position and orientation of the **W-CS** are defined by basic transformations in the preset table.

Further information: "Preset management", Page 1025



Transformations in the workpiece coordinate system (W-CS)

HEIDENHAIN recommends using the following transformations in the workpiece coordinate system **W-CS**:

- **TRANS DATUM** function before tilting the working plane

  Further information: "Datum shift with TRANS DATUM", Page 1046
- Function **TRANS MIRROR** or Cycle **8 MIRRORING** before tilting the working plane with spatial angles

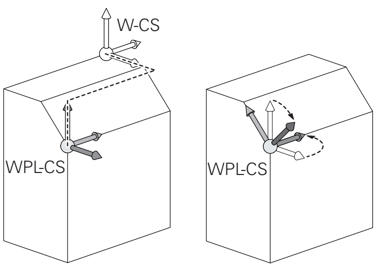
  Further information: "Mirroring with TRANS MIRROR", Page 1047

  Further information: "Cycle 8 MIRRORING", Page 1036
- **PLANE** functions for tilting the working plane (option 8)

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 1054

> You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

With these transformations, the position and orientation of the working plane coordinate system **WPL-CS** are changed.



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

In the machine parameter **planeOrientation** (no. 201202) the machine manufacturer defines whether the control interprets input values of Cycle **19 WORKING PLANE** as spatial angles or as axis angles.

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.

Additional transformations with Global Program Settings (GPS, option 44)

In the **GS** workspace (option 44) you can define additional transformations in the workpiece coordinate system **W-CS**:

- **Additive basic rotat. (W-CS)**

  The effects of this function are added to a basic rotation or a 3D basic rotation from the preset table or the pallet preset table. This function is the first transformation that is possible in the **W-CS**.
- **Shift (W-CS)**

  This function is in effect in addition to a datum shift defined in the NC program with the **TRANS DATUM** function and before the working plane is tilted.
- **Mirroring (W-CS)**

  The function is active in addition to a mirror image (function **TRANS MIRROR** or Cycle **8 MIRRORING**) defined in the NC program and before tilting the working plane.
- **Shift (mW-CS)**

  This function is in effect in the modified workpiece coordinate system. This function is in effect after the **Shift (W-CS)** and **Mirroring (W-CS)** functions and before the working plane is tilted.

Further information: "Globale Programmeinstellungen GPS", Page

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.

  Further information: "Input coordinate system I-CS", Page 1021
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.

  Further information: "Working plane coordinate system WPL-CS", Page 1018
- The result of transformations built upon each other depends on the programming sequence.

16.1.6 Working plane coordinate system WPL-CS

Application

In the working plane coordinate system **WPL-CS** you define the position and orientation of the input coordinate system **I-CS** and therefore the reference for the coordinate system in the NC program. You do this by programming transformations after having tilted the working plane.

Further information: "Input coordinate system I-CS", Page 1021

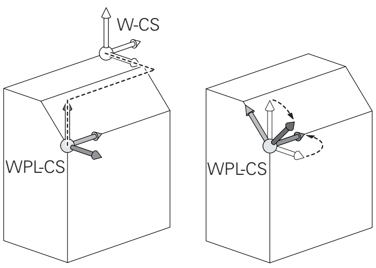
Description of function

Properties of the working plane coordinate system WPL-CS

The working plane coordinate system **WPL-CS** is a three-dimensional Cartesian coordinate system. You use transformations in the workpiece coordinate system **W-CS** to define the coordinate origin of the **WPL-CS**.

Further information: "Workpiece coordinate system W-CS", Page 1016

If no transformations are defined in the **W-CS**, then the position and orientation of the **W-CS** and **WPL-CS** are identical.



Transformations in the working plane coordinate system WPL-CS

HEIDENHAIN recommends using the following transformations in the working plane coordinate system **WPL-CS**:

- **TRANS DATUM** function

  Further information: "Datum shift with TRANS DATUM", Page 1046
- **TRANS MIRROR** or Cycle **8 MIRRORING** function

  Further information: "Mirroring with TRANS MIRROR", Page 1047

  Further information: "Cycle 8 MIRRORING", Page 1036
- **TRANS ROTATION** function or Cycle **10 ROTATION**

  Further information: "Rotations with TRANS ROTATION", Page 1050

  Further information: "Cycle 10 ROTATION ", Page 1038
- **TRANS SCALE** function or Cycle **11 SCALING FACTOR**

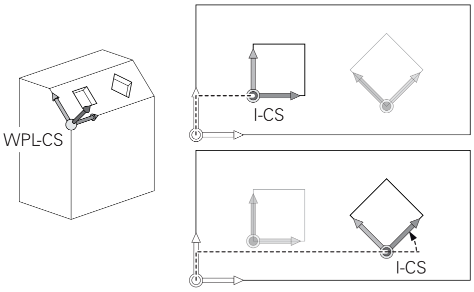
  Further information: "Scaling with TRANS SCALE", Page 1051

  Further information: "Cycle 11 SCALING FACTOR ", Page 1040
- Cycle **26 AXIS-SPECIFIC SCALING**

  Further information: "Cycle 26 AXIS-SPECIFIC SCALING ", Page 1041
- **PLANE RELATIV** function (option 8)

  Further information: "PLANE RELATIV", Page 1080

With these transformations you modify the position and orientation of the input coordinate system **I-CS**.



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

Additional transformations with Global Program Settings (GPS, option 44)

The **Rotation (WPL-CS)** transformation in the **GS** workspace has an additive effect to a rotation in the NC program.

Further information: "Global Program Settings (GPS, option 44)", Page 1217

Additional transformations with mill-turning (option 50)

The following additional transformations are available with the mill-turning software option:

- Precession angle with the following cycles:
  - Cycle **800 ADJUST XZ SYSTEM**
  - Cycle **801 RESET ROTARY COORDINATE SYSTEM**
  - Cycle **880 GEAR HOBBING**
- OEM transformations defined by machine manufacturers for special turning kinematics

> Machine manufacturers can also define an OEM transformation and a precession angle without software option 50.
>
> An OEM transformation takes effect before the precession angle.
>
> If an OEM transformation or a precession angle is defined, the control shows the values on the **POS** tab of the **Status** workspace. These transformations are also in effect in milling mode!
>
> **Further information:** "POS tab", Page 179

Additional transformation with Gear Cutting (option 157)

You can use the following cycles to define a precession angle:

- Cycle **286 GEAR HOBBING**
- Cycle **287 GEAR SKIVING**

> Machine manufacturers can also define a precession angle without Gear Cutting (software option 157)

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.

  Further information: "Input coordinate system I-CS", Page 1021
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.
- The result of transformations built upon each other depends on the programming sequence.
- As a **PLANE** function (option 8), **PLANE RELATIV** has an effect in the workpiece coordinate system **W-CS** and orients the working plane coordinate system **WPL-CS**. The values of additive tilting always relate to the current **WPL-CS**.

16.1.7 Input coordinate system I-CS

Application

The programmed values in the NC program refer to the input coordinate system **I-CS**. You use positioning blocks to program the position of the tool.

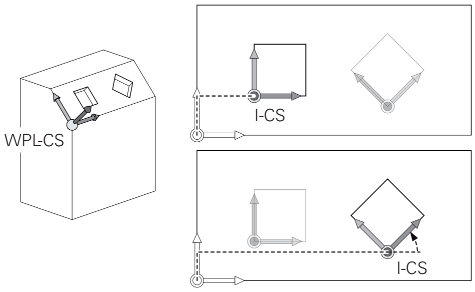
Description of function

Properties of the input coordinate system I-CS

The input coordinate system **I-CS** is a three-dimensional Cartesian coordinate system. You use transformations in the working plane coordinate system **WPL-CS** to define the coordinate origin of the **I-CS**.

Further information: "Working plane coordinate system WPL-CS", Page 1018

If no transformations are defined in the **WPL-CS**, then the position and orientation of the **WPL-CS** and **I-CS** are identical.



Positioning blocks in the input coordinate system I-CS

In the input coordinate system **I-CS** you use positioning blocks to define the position of the tool. The position of the tool defines the position of the tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 1022

You can define the following positioning blocks:

- Paraxial positioning blocks
- Path functions with Cartesian or polar coordinates
- Straight lines **LN** with Cartesian coordinates and surface normal vectors (option 9)
- Cycles

11 X+48 R+	; Paraxial positioning block
11 L X+48 Y+102 Z-1.5 R0	; Path function **L**
11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 R0	; Straight line **LN** with Cartesian coordinates and surface normal vector

Position display

The following modes of the position display are referenced to the input coordinate system **I-CS**:

- **Nominal pos. (NOML)**
- **Actual pos. (ACT)**

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.

  Further information: "Working plane coordinate system WPL-CS", Page 1018

16.1.8 Tool coordinate system T-CS

Application

In the tool coordinate system **T-CS** the control implements tool compensations and tool inclinations.

Description of function

Properties of the tool coordinate system T-CS

The tool coordinate system **T-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the tool tip TIP.

You make entries in the tool management to define the tool tip relative to the tool carrier reference point. The machine manufacturer usually defines the tool carrier reference point on the spindle tip.

Further information: "Presets in the machine", Page 210

You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

- **L**
- **DL**
- **ZL** (option 50, option 156)
- **XL** (option 50, option 156)
- **YL** (option 50, option 156)
- **DZL** (option 50, option 156)
- **DXL** (option 50, option 156)
- **DYL** (option 50, option 156)
- **LO** (option 156)
- **DLO** (option 156)

Further information: "Tool carrier reference point", Page 271

You use positioning blocks in the input coordinate system **I-CS** to define the position of the tool and therefore the position of the **T-CS**.

Further information: "Input coordinate system I-CS", Page 1021

You can use miscellaneous functions to also program in other reference systems, such as **M91** for the machine coordinate system **M-CS**.

Further information: "Traversing in the machine coordinate system M-CS with M91", Page 1322

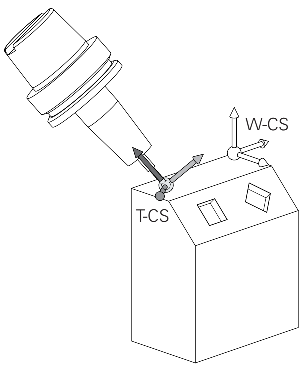
The orientation of the **T-CS** in most cases is identical to that of the **I-CS**.

If the following functions are active, the orientation of the **T-CS** depends on the tool angle of inclination:

- Miscellaneous function **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1341
- **PLANE RELATIV** function (option 9)

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104



Use the miscellaneous function **M128** to define the tool angle of inclination in the machine coordinate system **M-CS** using axis angles. The effects of the tool angle of inclination depend on the machine kinematics:

Further information: "Notes", Page 1344

11 L X+10 Y+45 A+10 C+5 R0 M128	; Straight line with miscellaneous function **M128** and axis angles

You can also define a tool angle of inclination in the working plane coordinate system **WPL-CS**, for example with **FUNCTION TCPM** or a straight line **LN**.

11 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	; **FUNCTION TCPM** with spatial angles
12 L A+0 B+45 C+0 R0 F2500	

11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 TX-0.08076201 TY-0.34090025 TZ0.93600126 R0 M128	; Straight line **LN** with surface normal vector and tool orientation

Transformations in the tool coordinate system T-CS

The following tool compensations have an effect in the tool coordinate system **T-CS**:

- Compensation values from the tool management

  Further information: "Tool compensation for tool length and tool radius", Page 1112
- Compensation values from the tool call

  Further information: "Tool compensation for tool length and tool radius", Page 1112
- Values of the compensation tables ***.tco**

  Further information: "Tool compensation with compensation tables", Page 1120
- Values of **FUNCTION TURNDATA CORR T-CS** (option 50)

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1124
- 3D tool compensation with surface normal vectors (option 9)

  Further information: "3D tool compensation (option 9)", Page 1126
- 3D tool radius compensation depending on the tool's contact angle using compensation-value tables (option 92)

  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1140

Position display

The display of the virtual tool axis **VT** refers to the tool coordinate system **T-CS**.

The control shows the values of **VT** in the **GS workspace** (option 44) and on the **GS** tab of the **Status** workspace.

Further information: "Global Program Settings (GPS, option 44)", Page 1217

The HR 520 and HR 550 FS handwheels show the values of **VT** in the display.

Further information: "Contents of an electronic handwheel display", Page 2072

16.2 Preset management

Application

The preset management allows setting and activating single presets. The presets to be saved may include, for example, the position and the misalignment of a workpiece in the preset table. The active row of the preset table serves as the workpiece preset in the NC program and as the origin of workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 210

Use the preset management in the following cases:

- To tilt the working plane of a machine with table or head rotation axes (option 8)
- To work on a machine with a head change system
- To machine several workpieces that are clamped down at different misaligned positions
- If REF-based datum tables were used on previous control models

Related topics

- Contents of preset table, write protection

  Further information: "Preset table", Page 2035

Description of function

Setting presets

Presets can be set in the following ways:

- Setting axis positions manually

  Further information: "Setting a preset manually", Page 1028
- Touch probe cycles in the **Setup** application

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557
- Touch probe cycles in the NC program

  Further information: "Programmable Touch Probe Cycles", Page 1589

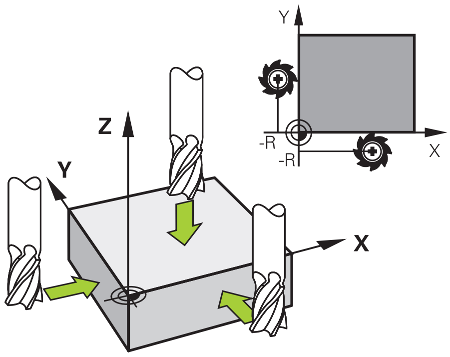
  Further information: "Cycle 247 PRESETTING ", Page 1042

If you try to write a value in a write-protected preset table row, the control cancels this process with an error message. Write-protection for this row must be rescinded first.

Further information: "Removing write protection", Page 2041

Setting a preset with milling cutters

If no workpiece touch probe is available, the preset can also be set by using a milling cutter. In this case, the values are not obtained by probing, but by scratching.



When scratching with a milling cutter, the tool is slowly moved toward the workpiece edge in the **Manual operation** application while the spindle is rotating.

As soon as the tool produces chips on the workpiece, the preset is manually set in the desired axis.

Further information: "Setting a preset manually", Page 1028

Activating presets

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

- Before activating a preset, check whether all columns contain values.

Presets can be activated in the following ways:

- Activating manually in the **Tables** operating mode

  Further information: "Activating a preset manually", Page 1029
- Cycle **247 PRESETTING**

  Further information: "Cycle 247 PRESETTING ", Page 1042
- **PRESET SELECT** function

  Further information: "Activating the preset with PRESET SELECT", Page 1030

When activating a preset, the control resets the following transformations:

- Datum shift with the **TRANS DATUM** function
- Mirror image with the **TRANS MIRROR** function or Cycle **8 MIRRORING**
- Rotation with the **TRANS ROTATION** function or Cycle **10 ROTATION**
- Scaling with the **TRANS SCALE** function or Cycle **11 SCALING FACTOR**
- Axis-specific scaling with Cycle **26 AXIS-SPECIFIC SCALING**

Tilting the working plane by using **PLANE** functions or Cycle **19 WORKING PLANE** will not be reset by the control.

Basic rotation and 3D basic rotation

The **SPA**, **SPB** and **SPC** columns define a spatial angle for orienting the workpiece coordinate system **W-CS**. This spatial angle defines the basic rotation or 3D basic rotation of the preset.

Further information: "Workpiece coordinate system W-CS", Page 1016

When a rotation around the tool axis is defined, the preset contains a basic rotation (e.g., **SPC** for tool axis **Z**). If one of the remaining columns is defined, the preset contains a 3D basic rotation. If the workpiece preset contains a basic rotation or 3D basic rotation, the control takes these values into account when executing an NC program.

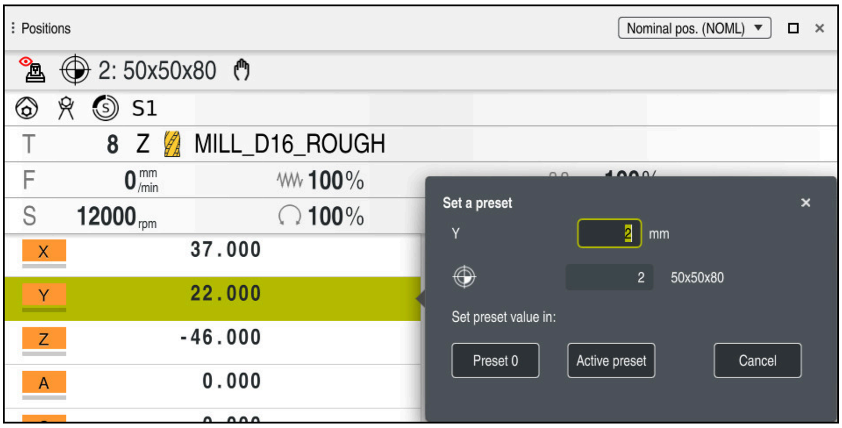
The **3D ROT** button (option 8) allows defining that the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application as well.

Further information: "3-D rotation window (option 8)", Page 1098

When a basic rotation or 3D basic rotation is active, the control displays a symbol in the **Positions** workspace.

Further information: "Active functions", Page 166

16.2.1 Setting a preset manually

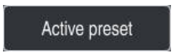


Set a preset window in the **Positions** workspace

When setting the preset manually, the values can be written either in row 0 of the preset table or in the active row.

To set a preset manually in an axis:

- Select the **Manual operation** application in the **Manual** operating mode
- Open the **Positions** workspace
- Traverse the tool to the desired position (e.g., for scratching)
- Select the row of the desired axis
- > The control opens the **Set a preset** window.
- Enter the value of the current axis position, relating to the new preset (e.g., **0**)
- > The control activates the **Preset 0** and **Active preset** buttons for selection.



- Select an option (e.g., **Active preset**)
- > The control saves the value in the selected preset table row and closes the **Set a preset** window.
- > The control updates the values in the **Positions** workspace.

> ℹ
> - The **Set the preset** button in the function bar opens the **Set a preset** window for the row marked in green.
> - When selecting **Preset 0**, the control automatically activates row 0 of the preset table as the workpiece preset.

16.2.2 Activating a preset manually

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

- Before activating a preset, check whether all columns contain values.

To activate a preset manually:

- Select the **Tables** operating mode
- Select the **Presets** application
- Select the desired row

Activate the preset

- Select **Activate the preset**
- > The control activates the preset.
- > The control displays the number and comment of the active preset in the **Positions** workspace and in the status overview.

Further information: "Description of function", Page 163

Further information: "Status overview on the TNC bar", Page 169

Notes

- The machine manufacturer uses the optional machine parameter **initial** (no. 105603) to define a default value for each column of a new row.
- In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (option 8). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.

  Further information: "3-D rotation window (option 8)", Page 1098
  In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- When scratching a workpiece with the radius of a milling cutter, the radius value must be taken into account in the preset.
- Even if the current preset contains a basic rotation or a 3D basic rotation, the **PLANE RESET** function will position the rotary axes at 0° in the **MDI** application.

  Further information: "The MDI Application ", Page 1933
- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

  Further information: "Pallet preset table", Page 1951

16.3 NC functions for preset management

16.3.1 Overview

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

16.3.2 Activating the preset with PRESET SELECT

Application

The **PRESET SELECT** function allows you to use a preset defined in the preset table and activate it as a new preset.

Requirement

- The preset table contains values
  Further information: "Preset management", Page 1025
- Workpiece preset has been defined
  Further information: "Setting a preset manually", Page 1028

Description of function

To activate the preset, use the preset number or the entry in the **Doc** column. If the entry in the **Doc** column is not unique, the control will activate the preset with the smallest preset number.

The **KEEP TRANS** syntax element allows defining that the control retains the transformations below:

- the **TRANS DATUM** function
- Cycle **8 MIRRORING** and the **TRANS MIRROR** function
- Cycle **10 ROTATION** and the **TRANS ROTATION** function
- Cycle **11 SCALING FACTOR** and the **TRANS SCALE** function
- Cycle **26 AXIS-SPECIFIC SCALING**

Input

11 PRESET SELECT #3 KEEP TRANS WP	; Activate row 3 of the table as the workpiece preset and maintain transformations

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET SELECT	Syntax initiator for activating a preset
#, **" "** or **QS**	Select the row of the preset table Fixed or variable number or name The row can be selected from a selection menu. For names the control only displays the rows in the preset table where the **Doc** column is defined.
KEEP TRANS	Retain simple transformations Optional syntax element
WP or **PAL**	Activate the preset for the workpiece or pallet Optional syntax element

Note

If you program **PRESET SELECT** without optional parameters, then the behavior is identical to Cycle **247 PRESETTING**.

Further information: "Cycle 247 PRESETTING ", Page 1042

16.3.3 Copying the preset with PRESET COPY

Application

The function **PRESET COPY** allows you to copy a preset defined in the preset table and activate the preset copied.

Requirement

- The preset table contains values
  Further information: "Preset management", Page 1025
- Workpiece preset has been defined
  Further information: "Setting a preset manually", Page 1028

Description of function

To select the preset to be copied, use the preset number or the entry in the **Doc** column. If the entry in the **Doc** column is not unique, the control will select the preset with the smallest preset number.

Input

11 PRESET COPY #1 TO #3 SELECT TARGET KEEP TRANS	; Copy row 1 of the preset table to row 3, activate row 3 as the workpiece preset and maintain transformations

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET COPY	Syntax initiator for copying and activating a workpiece preset
#, **" "** or **QS**	Select the row of the preset table to be copied Fixed or variable number or name The row can be chosen from a selection menu. With names, the control displays in the selection menu only the rows of the preset table for which the **Doc** column is defined.
TO #, **" "** or **QS**	Select the new row of the preset table Fixed or variable number or name The row can be chosen from a selection menu. With names, the control displays in the selection menu only the rows of the preset table for which the **Doc** column is defined.
SELECT TARGET	Activate the copied row of the preset table as the workpiece preset Optional syntax element
KEEP TRANS	Optional syntax element

16.3.4 Correcting the preset with PRESET CORR

Application

The function **PRESET CORR** allows you to correct the active preset.

Requirement

- The preset table contains values

  Further information: "Preset management", Page 1025
- Workpiece preset has been defined

  Further information: "Setting a preset manually", Page 1028

Description of function

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. When correcting the OFFS values, the values are referenced to the machine coordinate system **M-CS**.

Further information: "Reference systems", Page 1010

Input

11 PRESET CORR X+10 SPC+45	; Correct the workpiece preset in **X** by +10 mm and in **SPC** by +45°

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET CORR	Syntax initiator for correcting the workpiece preset
X, **Y**, **Z**	Compensation values in the principal axes Optional syntax element
SPA, **SPB**, **SPC**	Compensation values for the spatial angle Optional syntax element
X_OFFS, **Y_OFFS**, **Z_OFFS**, **A_OFFS**, **B_OFFS**, **C_OFFS**, **U_OFFS**, **V_OFFS**, **W_OFFS**	Compensation value for the offsets, referenced to the machine datum Optional syntax element

16.4 Datum table

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. The datums can be called from within an NC program, for example in order to execute machining processes on several workpieces at the same position. The active row of the datum table serves as the workpiece datum in the NC program.

Related topics

- Contents and creation of a datum table
  Further information: "Datum table", Page 2045
- Editing a datum table during a program run
  Further information: "Compensation during program run", Page 1974
- Preset table
  Further information: "Preset table", Page 2035

Description of function

The datums from a datum table are referenced to the current workpiece preset. The coordinate values from datum tables are only effective as absolute coordinate values.

Datum tables can be used in the following situations:

- Frequent use of the same datum shift
- Recurring machining sequences on different workpieces
- Recurring machining sequences at different positions on the workpiece

Activating the datum table manually

A datum table can be activated manually for the **Program Run** operating mode.

In the **Program Run** operating mode, the **Program settings** window contains the **Tables** area. In this area, a datum table and both compensation tables can be selected in one selection window for running the program.

When activating a table, the control will highlight this table with the status **M**.

16.4.1 Activating the datum table in the NC program

To activate a datum table in the NC program:

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **SEL TABLE**
- > The control opens the action bar.
- Select **Selection**
- > A file selection window opens.
- Select datum table
- Select **Select**

If the datum table is not stored in the same directory as the NC program, the complete path name must be defined. The **Program settings** window allows defining whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 220

> If you enter the datum table name manually, please note the following:
> - If the datum table is stored in the same directory as the NC program, enter the file name only.
> - If the datum table is not stored in the same directory as the NC program, enter the complete path.

Definition

File format	Definition
.d	Datum table

16.5 Coordinate transformation cycles

16.5.1 Fundamentals

Once a contour has been programmed, the control can execute it on the workpiece at various locations and in different sizes by using cycles for coordinate transformation.

Effectiveness of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

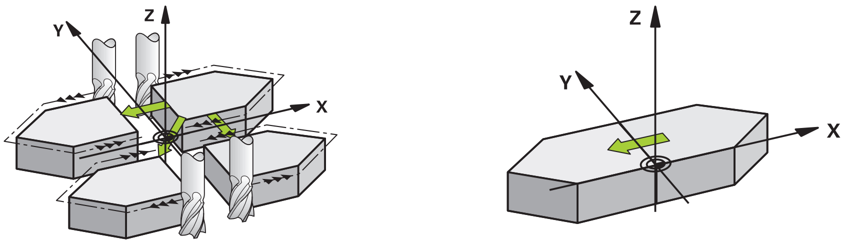
Reset coordinate transformation:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM NC block (these M functions depend on the machine parameters)
- Select a new NC program

16.5.2 Cycle 8 MIRRORING

ISO programming
G28

Application



The control can machine the mirror image of a contour in the working plane.

Mirroring becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. 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; this does not apply to SL cycles
- 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 1047

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

i	For working in a tilted system with Cycle **8**, the following procedure is recommended: ■ **First** program the tilting movement and **then** call Cycle **8 MIRRORING**!

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**

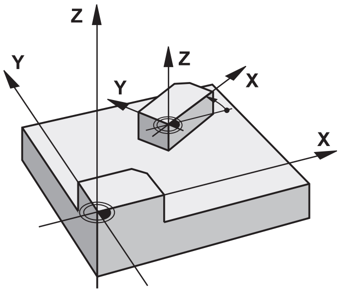
Example

```
11 CYCL DEF 8.0 MIRRORING
12 CYCL DEF 8.1 X Y Z
```

16.5.3 Cycle 10 ROTATION

ISO programming
G73

Application



Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

The ROTATION cycle becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. The active angle of rotation is shown in the additional status display.

Reference axis for the rotation angle:

- X/Y plane: X axis
- Y/Z plane: Y axis
- Z/X plane: Z axis

Reset

Program Cycle **10 ROTATION** again and specify a rotation angle of 0°.

Related topics

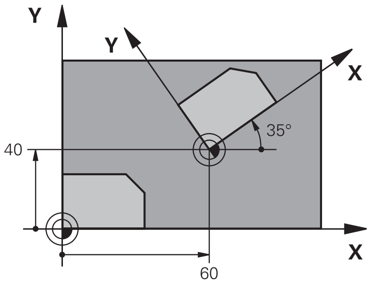
- Rotation with **TRANS ROTATION**

  Further information: "Rotations with TRANS ROTATION", Page 1050

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **10** cancels an active radius compensation. If necessary, reprogram the radius compensation.
- After defining Cycle **10**, move both axes of the working plane to activate the rotation for all axes.

Cycle parameters

Help graphic	Parameter
	**Rotation angle?** Enter the angle of rotation in degrees (°). Enter the value as an incremental or absolute value. Input: **-360.000...+360.000**

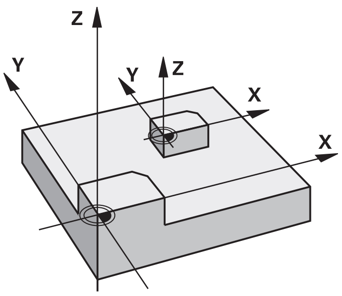
Example

```
11 CYCL DEF 10.0 ROTATION
12 CYCL DEF 10.1  ROT+35
```

16.5.4 Cycle 11 SCALING FACTOR

ISO programming
G72

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 becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. 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.

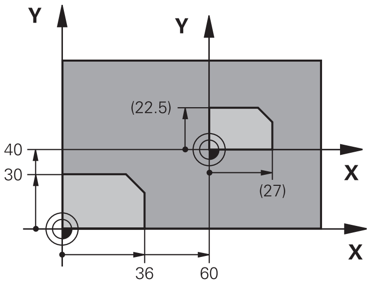
Reset

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 1051

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

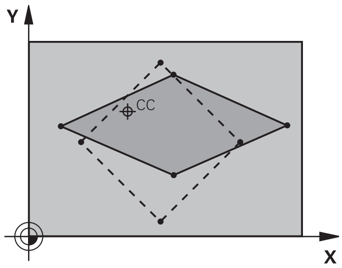
```
11 CYCL DEF 11.0 SCALING FACTOR
12 CYCL DEF 11.1 SCL 0.75
```

16.5.5 Cycle 26 AXIS-SPECIFIC SCALING

ISO programming

NC syntax is available only in Klartext programming.

Application



Use Cycle **26** to account for shrinkage and allowance factors for each axis.

The scaling factor becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. 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.

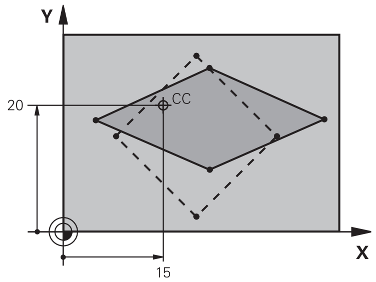
Notes

- 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

- Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.
- 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 the action bar. 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**

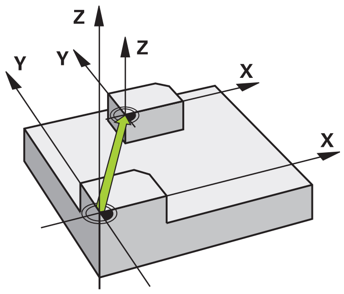
Example

```
11 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
12 CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```

16.5.6 Cycle 247 PRESETTING

ISO programming
G247

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 **Program Run** the control shows the active preset number behind the preset symbol in the **Positions** workspace.

Related topics

- Activate the preset
  Further information: "Activating the preset with PRESET SELECT", Page 1030
- Copy the preset
  Further information: "Copying the preset with PRESET COPY", Page 1031
- Correct the preset
  Further information: "Correcting the preset with PRESET CORR", Page 1032
- Setting and activating presets
  Further information: "Preset management", Page 1025

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- When activating a preset from the preset table, the control resets the datum shift, mirroring, rotation, 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** operating mode.
- Cycle **247** is also in effect in the simulation.

Cycle parameters

Help graphic	Parameter
	Number for preset? Enter the number of the desired preset from the preset table. Alternatively, you can use the button with the preset symbol in the action bar to directly select the desired preset from the preset table. Input: **0...65535**

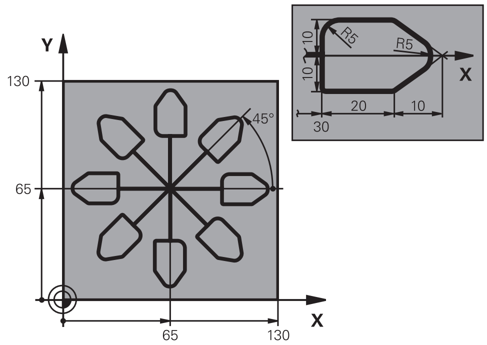
Example

```
11 CYCL DEF 247 PRESETTING ~
    Q339=+4                    ;PRESET NUMBER
```

16.5.7 Example: coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- Machining within a subprogram



0 BEGIN PGM C220 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+130 Y+130 Z+0	
3 TOOL CALL 1 Z S4500	; Tool call
4 L Z+100 R0 FMAX M3	; Retract the tool
5 TRANS DATUM AXIS X+65 Y+65	; Shift datum to center
6 CALL LBL 1	; Call milling operation
7 LBL 10	; Set label for program-section repeat
8 CYCL DEF 10.0 ROTATION	
9 CYCL DEF 10.1 IROT+45	
10 CALL LBL 1	; Call milling operation
11 CALL LBL 10 REP6	; Jump back to LBL 10; repeat six times
12 CYCL DEF 10.0 ROTATION	
13 CYCL DEF 10.1 ROT+0	
14 TRANS DATUM RESET	; Reset datum shift
15 L Z+250 R0 FMAX	; Retract the tool
16 M30	; End program
17 LBL 1	; Subprogram 1
18 L X+0 Y+0 R0 FMAX	; Define milling operation
19 L Z+2 R0 FMAX	
20 L Z-5 R0 F200	
21 L X+30 RL	
22 L IY+10	
23 RND R5	
24 L IX+20	
25 L IX+10 IY-10	
26 RND R5	
27 L IX-10 IY-10	
28 L IX-10 IY-10	

```
29 L IX-20
30 L IY+10
31 L  X+0  Y+0 R0 F5000
32 L  Z+20 R0 FMAX
33 LBL 0
34 END PGM C220 MM
```

16.6 NC functions for coordinate transformation

16.6.1 Overview

The control provides the following **TRANS** functions:

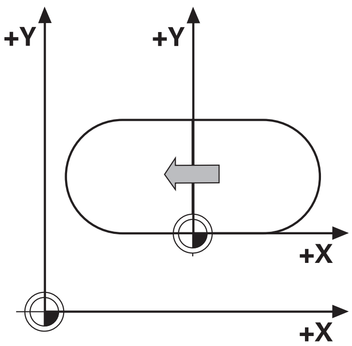
Syntax	Function	Further information
TRANS DATUM	Shift the workpiece datum	Page 1046
TRANS MIRROR	Mirror an axis	Page 1047
TRANS ROTATION	Rotation about the tool axis	Page 1050
TRANS SCALE	Scale contours and positions	Page 1051

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 1021



Related topics

- Coordinate transformation cycles

  Further information: "Coordinate transformation cycles", Page 1034
- **PLANE** functions (option 8)

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 1054
- Reference systems

  Further information: "Reference systems", Page 1010

16.6.2 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

- Contents of the datum table

  Further information: "Datum table", Page 2045
- Activating the datum table

  Further information: "Activating the datum table in the NC program", Page 1034
- Machine presets

  Further information: "Presets in the machine", Page 210

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.

The control displays the result of the datum shift in the **Positions** workspace.

Further information: "Positions workspace", Page 163

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 the NC program", Page 1034

The control displays the datum shift and the path to the datum table on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 181

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

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
" " or **QS**	Path to the datum table Fixed or variable name Optional syntax element Only if **TABLE** has been selected

Notes

- The **TRANS DATUM** function replaces Cycle **7 DATUM SHIFT**. If you import an NC program from an older control, then, during editing, the control turns Cycle **7** into the **TRANS DATUM** NC function.
- If you execute an absolute data 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.
- Absolute values reference the workpiece preset. Incremental values reference the workpiece datum.

  Further information: "Presets in the machine", Page 210
- In machine parameter **transDatumCoordSys** (no. 127501), the machine manufacturer defines the reference system referred to by the values in the position display.

  Further information: "Reference systems", Page 1010

16.6.3 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 the mirroring.

Related topics

- Cycle **8 MIRRORING**

  Further information: "Cycle 8 MIRRORING", Page 1036
- Additive mirroring within the global program settings GPS (option 44)

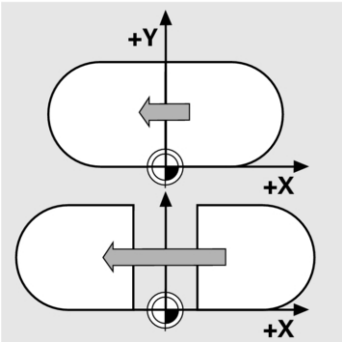
  Further information: "Function Mirroring (W-CS)", Page 1223

Description of function

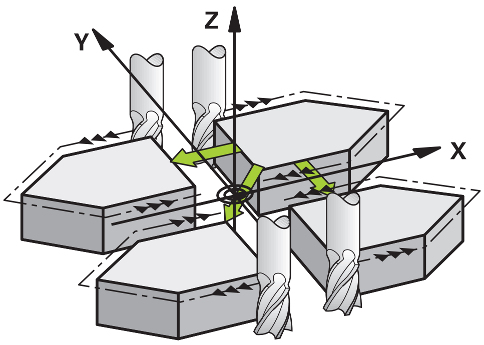
Mirroring is a modal function that in effect as soon as it has been defined in the NC program.

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.

Further information: "Presets in the machine", Page 210



If you mirror only one axis, the machining direction of the tool is reversed. The rotational direction defined in a cycle will remain unchanged, such as when defined within one of the OCM cycles (option 167).

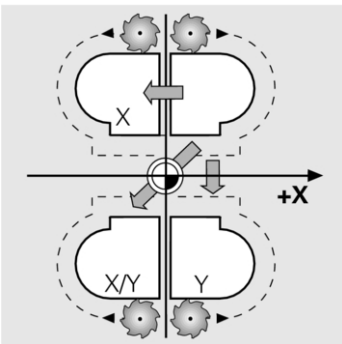


Depending on the selected **AXIS** axis values, the control will mirror the following working planes:

- **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 208

You can select up to three axis values.



If mirroring is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 181

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 be used only in the **FUNCTION MODE MILL** machining mode.
  Further information: "Switching the operating mode with FUNCTION MODE", Page 234
- If you execute mirroring with **TRANS MIRROR** or Cycle **8 MIRRORING**, then the control overwrites the current mirroring.
  Further information: "Cycle 8 MIRRORING", Page 1036

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 1016

16.6.4 Rotations with TRANS ROTATION

Application

With the **TRANS ROTATION** function, you can rotate contours or positions around a rotation angle.

The **TRANS DATUM RESET** function allows you to reset the rotation.

Related topics

- Cycle **10 ROTATION**
  Further information: "Cycle 10 ROTATION ", Page 1038
- Additive rotation within the global program settings (GPS, option 44)

Description of function

Rotation is a modal function that is in effect as soon as it has been defined in the NC program.

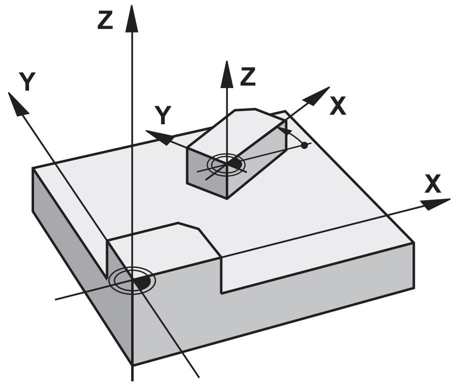
The control rotates machining in the working plane about the active workpiece datum.

Further information: "Presets in the machine", Page 210

The control rotates the input coordinate system (**I-CS**) as follows:

- Based on the angle reference axis, i.e. the main axis
- About the tool axis

Further information: "Designation of the axes on milling machines", Page 208



A rotation can be programmed as follows:

- Absolute, relative to the positive main axis
- Incremental, relative to the last active rotation

If rotation is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 181

Input

11 TRANS ROTATION ROT+90	; Rotate machining by 90°

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS ROTATION	Start of syntax for a rotation
ROT or **RESET**	Enter an absolute or incremental angle of rotation or reset rotation Fixed or variable number

Notes

- This function can be used only in the **FUNCTION MODE MILL** machining mode.
  Further information: "Switching the operating mode with FUNCTION MODE", Page 234
- If you execute an absolute rotation with **TRANS ROTATION** or Cycle **10 ROTATION**, then the control overwrites the values of the current rotation. The control adds the incremental values to the values of the current rotation.
  Further information: "Cycle 10 ROTATION ", Page 1038

16.6.5 Scaling with TRANS SCALE

Application

The **TRANS SCALE** 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 the scaling.

Related topics

- Cycle **11 SCALING FACTOR**
  Further information: "Cycle 11 SCALING FACTOR ", Page 1040

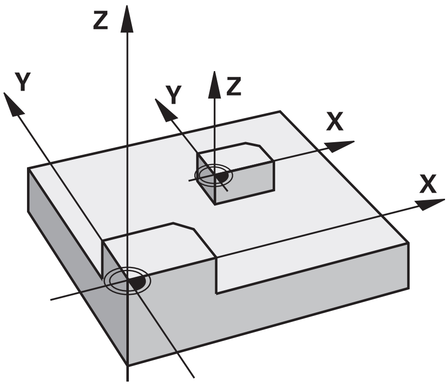
Description of function

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.

Further information: "Presets in the machine", Page 210



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 **Status** workspace.

Further information: "TRANS tab", Page 181

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 be used only in the **FUNCTION MODE MILL** machining mode.
  Further information: "Switching the operating mode with FUNCTION MODE", Page 234
- 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: "Cycle 11 SCALING FACTOR ", Page 1040
- 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.

16.7 Tilting the working plane (option 8)

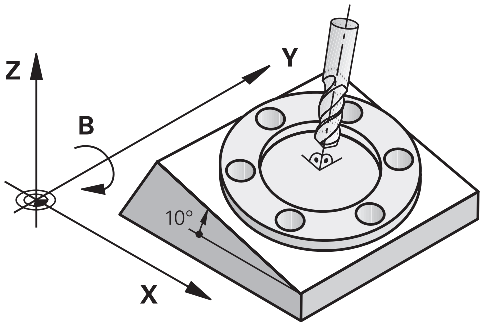
16.7.1 Fundamentals

Tilting the working plane of machines with rotary axes allows you to machine several workpiece sides in just one clamping setup, for example. The tilting functions also allow aligning a workpiece clamped at an incorrect angle.

The working plane can be tilted only when tool axis **Z** is active.

The control functions for tilting the working plane are coordinate transformations. The working plane is always perpendicular to the direction of the tool axis.

Further information: "Working plane coordinate system WPL-CS", Page 1018



There are two functions available for tilting the working plane:

- Manual tilting with the **3-D rotation** window in the **Manual operation** application
  Further information: "3-D rotation window (option 8)", Page 1098
- Tilting under program control with the **PLANE** functions in the NC program
  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 1054

> You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

Notes concerning different machine kinematics

When no transformations are active and the working plane is not tilted, the linear machine axes move in parallel with the basic coordinate system **B-CS**. In this process, machines behave almost identically, regardless of the kinematics.

Further information: "Basic coordinate system B-CS", Page 1014

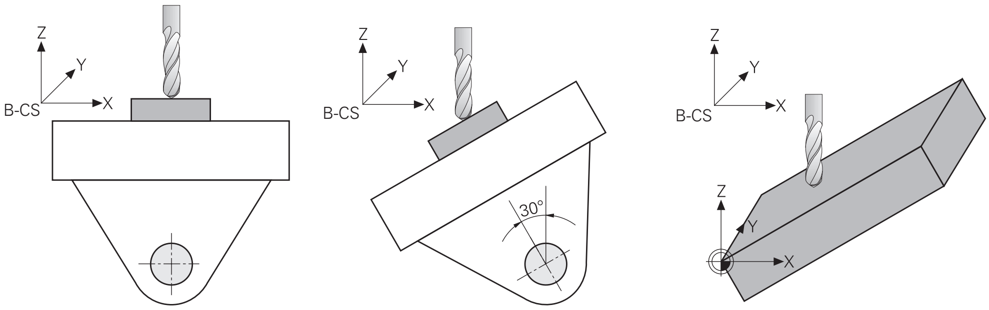
When tilting the working plane, the control moves the machine axes according to the kinematics.

Please observe the aspects below regarding the machine kinematics:

- Machine with table rotary axes

  With this kinematic model, the table rotary axes execute the tilting movement and the position of the workpiece in the work envelope changes. The linear machine axes move in the tilted working plane coordinate system **WPL-CS** just as they do in the non-tilted **B-CS**.

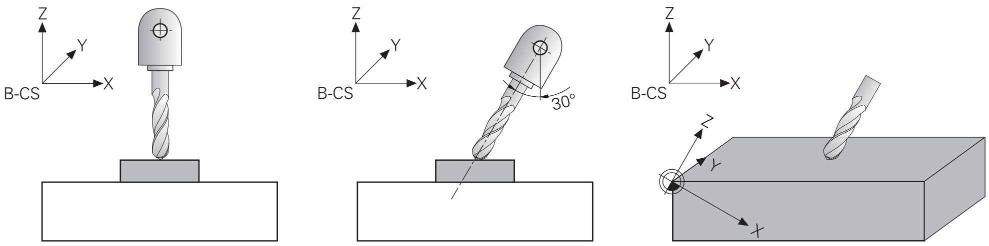
  Further information: "Working plane coordinate system WPL-CS", Page 1018



- Machine with head rotary axes

  With this kinematic model, the head rotary axes execute the tilting movement and the position of the workpiece in the work envelope remains the same. In the tilted **WPL-CS**, at least two linear machine axes no longer move in parallel with the non-tilted **B-CS**, depending on the rotary angle.

  Further information: "Working plane coordinate system WPL-CS", Page 1018



16.7.2 Tilting the working plane with PLANE functions (option 8)

Fundamentals

Application

Tilting the working plane of machines with rotary axes allows you to machine several workpiece sides in just one clamping setup, for example.

The tilting functions also allow aligning a workpiece clamped at an incorrect angle.

Related topics

- Machining types by number of axes

  Further information: "Types of machining according to number of axes", Page 1304
- Adopt tilted working plane in the **Manual** operating mode with the **3-D rotation** window

  Further information: "3-D rotation window (option 8)", Page 1098

Requirements

- Machine with rotary axes

  3+2 axes machining requires at least two rotary axes. Removable axes as an additional top table are also possible.
- Kinematics description

  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 1 (software option 8)
- Tool with tool axis **Z**

Description of function

Tilting the working plane defines the orientation of the working plane coordinate system **WPL-CS**.

Further information: "Reference systems", Page 1010

> The position of the workpiece datum and consequently the orientation of the working plane coordinate system **WPL-CS** can be defined by using the **TRANS DATUM** function before tilting the working plane in the workpiece coordinate system **W-CS**.
>
> A datum shift is always in effect in the active **WPL-CS**, meaning after the tilting function if applicable. If the workpiece datum is shifted for the tilting process, an active tilting function may have to be reset.
>
> **Further information:** "Datum shift with TRANS DATUM", Page 1046

In practice, workpiece drawings show different specified angles, which is why the control offers different **PLANE** functions with different options for defining angles.

Further information: "Overview of PLANE functions", Page 1056

In addition to the geometric definition of the working plane, every **PLANE** function allows specifying how the control positions the rotary axes.

Further information: "Rotary axis positioning", Page 1088

If the geometric definition of the working plane results in no unambiguous tilting position, the desired tilting solution can be selected.

Further information: "Tilting solution", Page 1091

Depending on the defined angles and the machine kinematics, there is a choice whether the control positions the rotary axes or orients the working plane coordinate system **WPL-CS** exclusively.

Further information: "Transformation types", Page 1095

Status display

Positions workspace

As soon as the working plane has tilted, the General status display in the **Positions** workspace contains an icon.

Further information: "Positions workspace", Page 163

> When deactivating or resetting the tilting function correctly, the icon indicating the tilted working plane must disappear.
>
> **Further information:** "PLANE RESET", Page 1084

Status workspace

When the working plane is tilted, the **POS** and **TRANS** tabs in the **Status** workspace contain information about the active orientation of the working plane.

When defining the working plane by using axis angles, the control displays the defined axis values. All alternative geometric definition options display the resulting spatial angles.

Further information: "POS tab", Page 179

Further information: "TRANS tab", Page 181

Overview of PLANE functions

The control provides the following **PLANE** functions:

Syntax element	Function	Further information
SPATIAL	Defines the working plane by means of three spatial angles	Page 1059
PROJECTED	Defines the working plane by means of two projection angles and one rotation angle	Page 1065
EULER	Defines the working plane by means of three Euler angles	Page 1069
VECTOR	Defines the working plane by means of two vectors	Page 1072
POINTS	Defines the working plane by means of the coordinates of three points	Page 1075
RELATIV	Defines the working plane by means of a single spatial angle with incremental effect	Page 1080
AXIAL	Defines the working plane by means of a maximum of three absolute or incremental axis angles	Page 1085
RESET	Resets tilting of the working plane	Page 1084

Notes

NOTICE

Danger of collision!

When the machine is switched on, the control tries to restore the switch-off status of the tilted plane. This is prevented under certain conditions. For example, this applies if axis angles are used for tilting while the machine is configured with spatial angles, or if you have changed the kinematics.

- If possible, reset tilting before shutting the system down
- Check the tilted condition when switching the machine back on

NOTICE

Danger of collision!

Cycle **8 MIRRORING** can have different effects in conjunction with the **Tilt working plane** function. The programming sequence, the mirrored axes, and the tilting function used are critical in this regard. There is a risk of collision during the tilting operation and subsequent machining!

- Check the sequence and positions using a graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Examples

1. When Cycle **8 MIRRORING** is programmed before the tilting function without rotary axes:
   - The tilt of the **PLANE** function used (except **PLANE AXIAL**) is mirrored
   - Mirroring takes effect after tilting with **PLANE AXIAL** or Cycle **19**
2. When Cycle **8 MIRRORING** is programmed before the tilting function with a rotary axis:
   - The mirrored rotary axis has no effect on the tilt specified in the **PLANE** function used, because only the movement of the rotary axis is mirrored

NOTICE

Danger of collision!

Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation.

- Make sure to retract the tool before changing the position of the rotary axis

- If you use the **PLANE** function when **M120** is active, the control automatically rescinds the radius compensation, which also rescinds the **M120** function.
- Always use **PLANE RESET** to cancel **PLANE** functions. Entering 0 in all **PLANE** parameters (e.g. all three spatial angles) exclusively resets the angles, but not the function.
- If you restrict the number of tilting axes with the **M138** function, your machine may provide only limited tilting possibilities. The machine manufacturer will decide whether the control takes the angles of deselected axes into account or sets them to 0.
- The control only supports tilting the working plane with spindle axis Z.

- You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

  If necessary, you can edit Cycle **19 WORKING PLANE**. However, you cannot insert the cycle again, because the control no longer offers the cycle for programming.

Tilting the working plane without rotary axes

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
> The machine manufacturer must take the precise angle into account (e.g., the angle of a mounted angle head in the kinematics description).

You can also orient the programmed working plane perpendicularly to the tool without defining rotary axes (e.g., when adapting the working plane for a mounted angle head).

Use the **PLANE SPATIAL** function and the **STAY** positioning behavior to swivel the working plane to the angle specified by the machine manufacturer.

Example of mounted angle head with permanent tool direction **Y**:

Example

```
11 TOOL CALL 5 Z S4500
12 PLANE SPATIAL SPA+0 SPB-90 SPC+0 STAY
```

> The tilt angle must be precisely adapted to the tool angle, otherwise the control will generate an error message.

PLANE SPATIAL

Application

Use the **PLANE SPATIAL** function to define the working plane by three spatial angles.

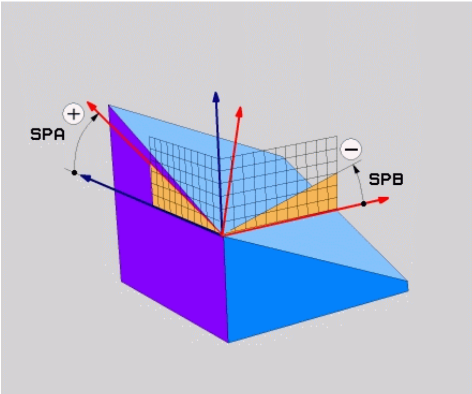
> Spatial angles are the most frequently used definition option for a working plane. The definition is not machine-specific, meaning that it is independent of the rotary axes actually present.

Related topics

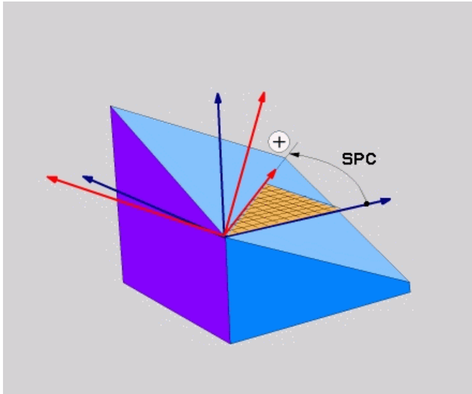
- Defining a single spatial angle with incremental effect
  Further information: "PLANE RELATIV", Page 1080
- Entering the axis angle
  Further information: "PLANE AXIAL", Page 1085

Description of function

Spatial angles define a working plane through three independent rotations in the workpiece coordinate system (**W-CS**), i. e. in the non-tilted working plane.



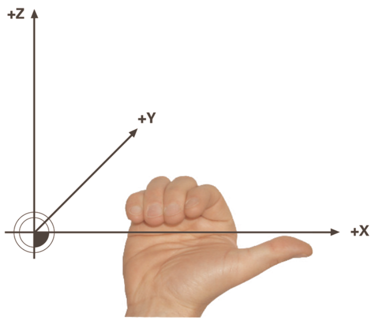
Spatial angles **SPA** and **SPB**



Spatial angle **SPC**

All three angles must be defined even if one or several angles equals 0.

As the spatial angles are programmed independently of the physically existing rotary axes, there is no need to differentiate between the head and the table axes as far as the signs are concerned. Always use the extended right-hand rule.



The thumb of your right hand points in the positive direction of the axis around which the rotation occurs. If you curl your fingers, the curled fingers point in the positive direction of rotation.

Entering the spatial angles as three independent rotations in the workpiece coordinate system **W-CS** in the programming sequence **A-B-C** is a challenge to many users. The challenge in particular is to take two coordinate systems into account simultaneously: the unmodified **W-CS** and the modified working plane coordinate system **WPL-CS**.

This is why the spatial angle can be alternatively defined by imagining three rotations layered on top of one another in the tilting sequence **C-B-A**. This alternative allows considering one coordinate system exclusively, meaning the modified working plane coordinate system **WPL-CS**.

Further information: "Notes", Page 1063

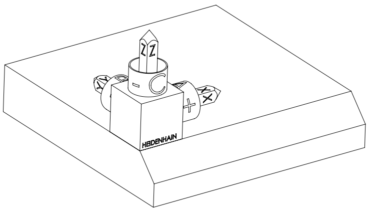
> This view equals three **PLANE RELATIV** functions programmed one-by-one, first with **SPC**, then with **SPB** and finally with **SPA**. The spatial angles with incremental effect **SPB** and **SPA** are referenced to the working plane coordinate system **WPL-CS**, i. e. to a tilted working plane.
>
> **Further information:** "PLANE RELATIV", Page 1080

Application example

Example

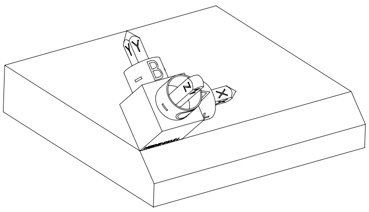
```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis



Using the defined spatial angle **SPA+45**, the control orients the tilted Z axis of **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **SPA** angle is around the non-tilted X axis.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following spatial angles:

- **SPA+45**, **SPB+0** and **SPC+90** for the second chamfer

  Further information: "Notes", Page 1063
- **SPA+45**, **SPB+0** and **SPC+180** for the third chamfer
- **SPA+45**, **SPB+0** and **SPC+270** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE SPATIAL	Defines the working plane by means of three spatial angles
SPA	Rotation around the X axis of the workpiece coordinate system **W-CS** Input: **-360.0000000...+360.0000000**
SPB	Rotation around the Y axis of the **W-CS** Input: **-360.0000000...+360.0000000**
SPC	Rotation around the Z axis of the **W-CS** Input: **-360.0000000...+360.0000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1091 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1095 Optional syntax element

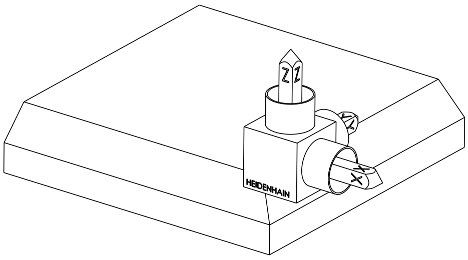
Notes

Comparison of views - Example: chamfer

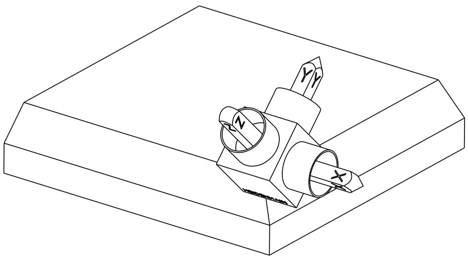
Example

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+90 TURN MB MAX FMAX SYM- TABLE ROT
```

View A-B-C



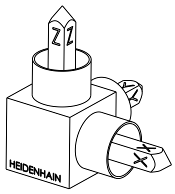
Initial state



SPA+45

Orientation of tool axis **Z**

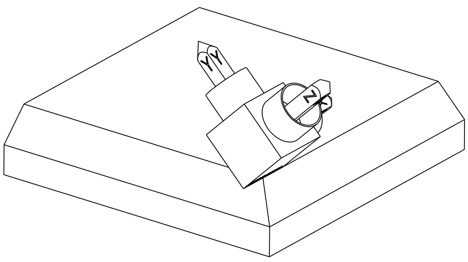
Rotation around the X axis of the non-tilted workpiece coordinate system **W-CS**



SPB+0

Rotation around the Y axis of the non-tilted **W-CS**

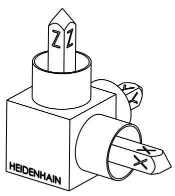
No rotation with value 0



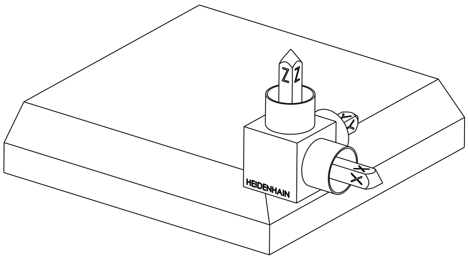
SPC+90

Orientation of main axis **X**

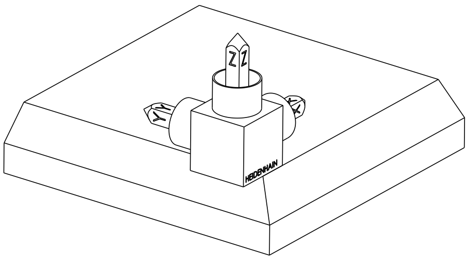
Rotation around the Z axis of the non-tilted **W-CS**



View C-B-A



Initial state



SPC+90

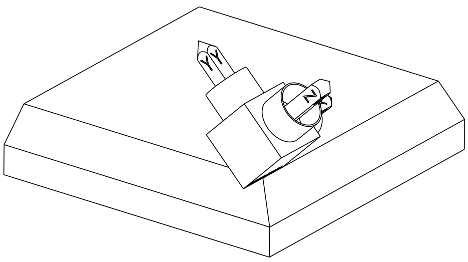
Orientation of main axis **X**

Rotation around the Z axis of the workpiece coordinate system **W-CS**, meaning in the non-tilted working plane

SPB+0

Rotation around the Y axis in the working plane coordinate system **WPL-CS**, meaning in the tilted working plane

No rotation with value 0



SPA+45

Orientation of tool axis **Z**

Rotation around the X axis in **WPL-CS**, meaning in the tilted working plane

Both views have an identical result.

Definition

Abbreviation	Definition
SP e. g. in **SPA**	Spatial

PLANE PROJECTED

Application

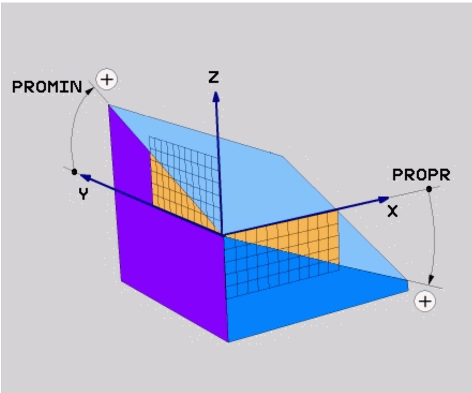
Use the **PLANE PROJECTED** function to define the working plane by two projection angles. Use an additional rotation angle to optionally align the X axis in the tilted working plane.

Description of function

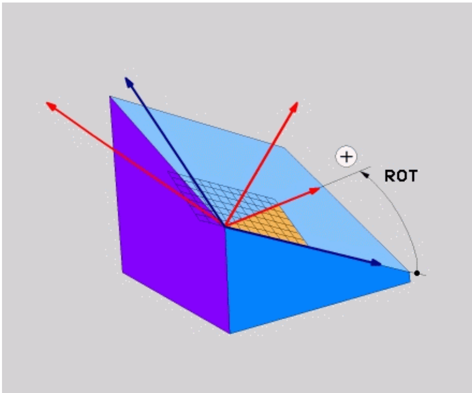
Projection angles define a working plane through two independent angles in the working planes **ZX** and **YZ** of the non-tilted working plane coordinate system **W-CS**.

Further information: "Designation of the axes on milling machines", Page 208

Use an additional rotation angle to optionally align the X axis in the tilted working plane.



Projection angles **PROMIN** and **PROPR**



Rotation angle **ROT**

All three angles must be defined even if one or several angles equals 0.

Entering the projection angles is easy for rectangular workpieces because the workpiece edges are the same as the projection angles.

The projection angles of non-rectangular workpieces can be obtained by imagining the working planes **ZX** and **YZ** as transparent panels with angle scales. When viewing the workpiece from the front through the **ZX** plane, the difference between the X axis and the workpiece edge equals the projection angle **PROPR**. Use the same procedure to obtain the projection angle **PROMIN** by viewing the workpiece from the left.

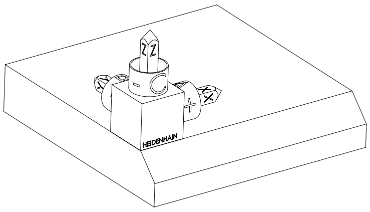
> When using **PLANE PROJECTED** for multi-side or internal machining, the hidden workpiece edges must be used or projected. Imagine the workpiece to be transparent in such cases.
>
> **Further information:** "Notes", Page 1068

Application example

Example

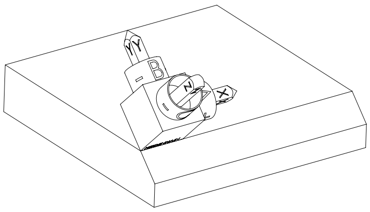
```
11 PLANE PROJECTED PROPR+0 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM- TABLE
ROT
```

Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis



Using the defined projection angle **PROMIN+45**, the control orients the Z axis of **WPL-CS** to be perpendicular with the chamfer surface. The angle from **PROMIN** is active in the working plane **YZ**.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following projection and rotation angles:

- **PROPR+45**, **PROMIN+0** and **ROT+90** for the second chamfer
- **PROPR+0**, **PROMIN-45** and **ROT+180** for the third chamfer
- **PROPR-45**, **PROMIN+0** and **ROT+270** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

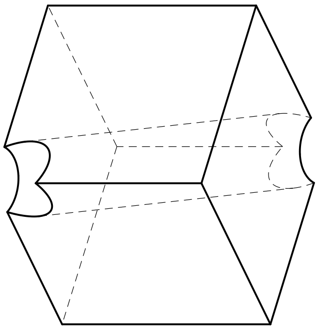
```
11 PLANE PROJECTED PROPR+0 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM- TABLE
ROT
```

The NC function includes the following syntax elements:

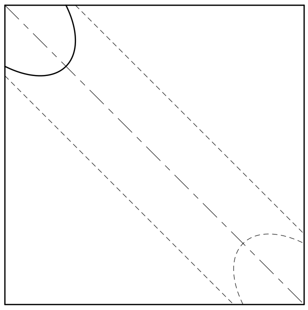
Syntax element	Meaning
PLANE PROJECTED	Syntax initiator for the working plane definition by means of two projection angles and one rotation angle
PROPR	Angle in working plane **ZX**, i. e. around the Y axis of the workpiece coordinate system **W-CS** Input: **-89.999999...+89.9999**
PROMIN	Angle in the working plane **YZ**, i. e. around the X axis of **W-CS** Input: **-89.999999...+89.9999**
ROT	Rotation around the Z axis of the tilted working plane coordinate system **WPL-CS** Input: **-360.0000000...+360.0000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1091 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1095 Optional syntax element

Notes

Procedure in case of hidden workpiece edges, using the example of a diagonal hole



Cube with a diagonal hole

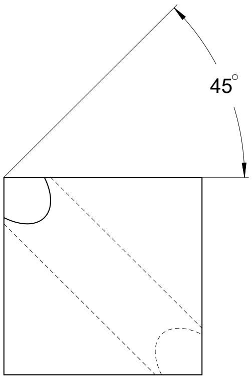


Front view, meaning projection on the **ZX** working plane

Example

```
11 PLANE PROJECTED PROPR-45  PROMIN+45 ROT+0 TURN MB MAX FMAX SYM-
   TABLE ROT
```

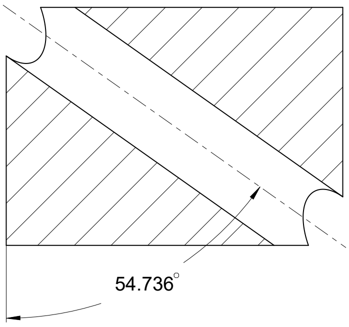
Comparison of projection and spatial angles



When imagining the workpiece to be transparent, the projection angles are easy to find.

Both projection angles are 45°.

> When defining the algebraic sign, ensure that the working plane is perpendicular to the center axis of the hole.



When defining the working plane by using spatial angles, the spatial diagonal must be considered.

The full section along the hole axis shows that the axis does not form an isosceles triangle with the lower and the left workpiece edge. This is why e. g. a spatial angle **SPA+45** produces an incorrect result.

Definition

Abbreviation	Definition
PROPR	Main plane
PROMIN	Minor plane
ROT	Angle of rotation

PLANE EULER

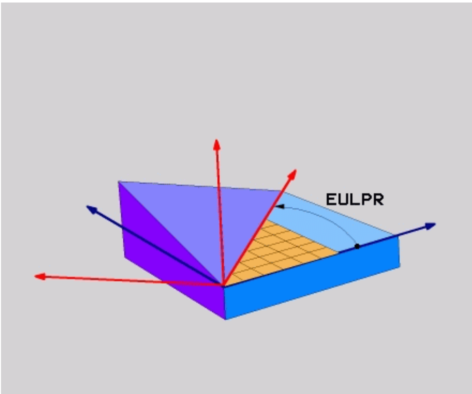
Application

Use the **PLANE EULER** function to define the working plane by three Euler angles.

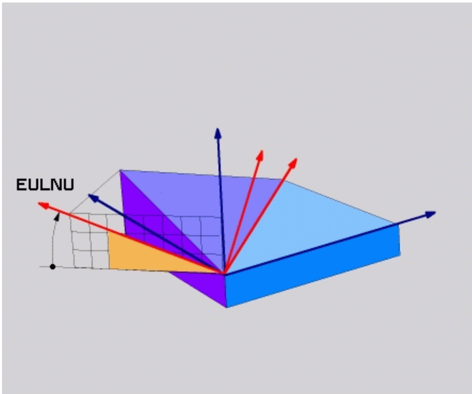
Description of function

Euler angles define a working plane as three rotations layered on top of one another, starting from the non-tilted workpiece coordinate system **W-CS**.

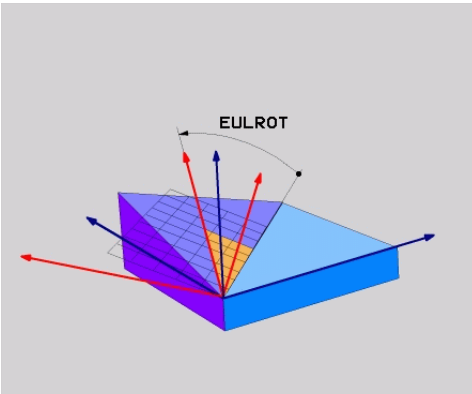
Use the third Euler angle to optionally align the tilted X axis.



Euler angle **EULPR**



Euler angle **EULNU**



Euler angle **EULROT**

All three angles must be defined even if one or several angles equals 0.

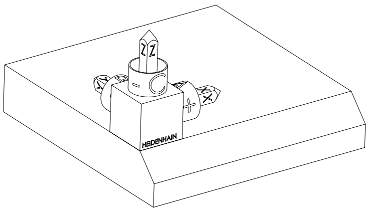
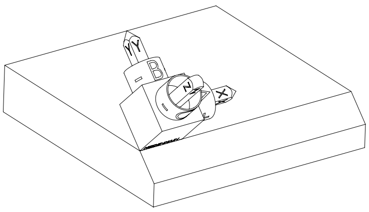
At first, the rotations layered on top of one another happen around the non-tilted Z axis, then around the tilted X axis and finally around the tilted Z axis.

> This view equals three **PLANE RELATIV** functions programmed one-by-one, first with **SPC**, then with **SPA** and finally with **SPC** again.
>
> **Further information:** "PLANE RELATIV", Page 1080
>
> The same result can be achieved by a **PLANE SPATIAL** function with the spatial angles **SPC** and **SPA**, followed by a rotation (e.g., with the **TRANS ROTATION** function).
>
> **Further information:** "PLANE SPATIAL", Page 1059
>
> **Further information:** "Rotations with TRANS ROTATION", Page 1050

Application example

Example

`11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT`

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the defined Euler angle **EULNU**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **EULNU** angle is around the non-tilted X axis. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following Euler angles:

- **EULPR+90**, **EULNU45** and **EULROT0** for the second chamfer
- **EULPR+180**, **EULNU45** and **EULROT0** for the third chamfer
- **EULPR+270**, **EULNU45** and **EULROT0** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

Example

```
11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE EULER	Syntax initiator for the working plane definition by means of three Euler angles
EULPR	Rotation around the Z axis of the workpiece coordinate system **W-CS** Input: **-180.000000...+180.000000**
EULNU	Rotation around the X axis of the tilted working plane coordinate system **WPL-CS** Input: **0...180.000000**
EULROT	Rotation around the Z axis of the tilted **WPL-CS** Input: **0...360.000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1091 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1095 Optional syntax element

Definition

Abbreviation	Definition
EULPR	Precession angle
EULNU	Nutation angle
EULROT	Angle of rotation

PLANE VECTOR

Application

Use the **PLANE VECTOR** function to define the working plane by two vectors.

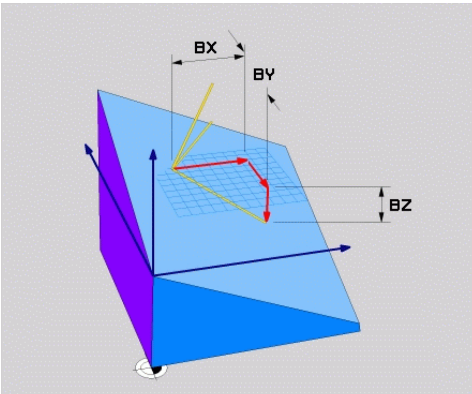
Related topics

- Output formats of NC programs

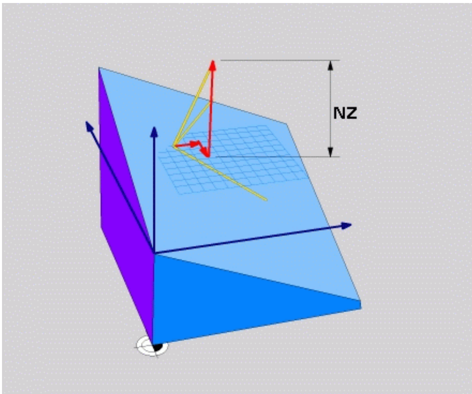
  Further information: "Output formats of NC programs", Page 1302

Description of function

Vectors define a working plane as two independent specifications of direction, starting from the non-tilted workpiece coordinate system **W-CS**.



Base vector with components **BX**, **BY** and **BZ**



NZ component of the normalized vector

All six components must be defined even if one or several components equals 0.

> There is no need to enter a normalized vector. The drawing dimensions or any values which will not alter the ratio between the components can be used.
>
> **Further information:** "Application example", Page 1073

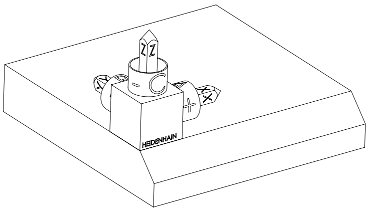
The base vector with components **BX**, **BY** and **BZ** defines the direction of the tilted X axis. The normal vector with components **NX**, **NY** and **NZ** defines the direction of the tilted Z axis and therefore indirectly the working plane. The normal vector is perpendicular to the tilted working plane.

Application example

Example

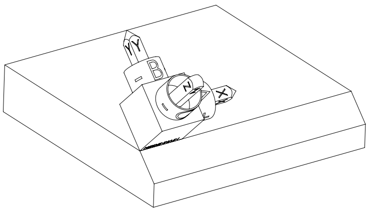
```
11 PLANE VECTOR BX+1 BY+0 BZ+0 NX+0 NY-1 NZ+1 TURN MB MAX FMAX SYM-
   TABLE ROT
```

Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis



Using the defined normal vector with the components **NX+0**, **NY-1** and **NZ+1**, the control orients the Z axis of the working plane coordinate system **WPL-CS** to be perpendicular with the chamfer surface.

The alignment of the tilted X axis equals the orientation of the non-tilted X axis due to component **BX+1**.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following vector components:

- **BX+0**, **BY+1** and **BZ+0** as well as **NX+1**, **NY+0** and **NZ+1** for the second chamfer
- **BX-1**, **BY+0** and **BZ+0** as well as **NX+0**, **NY+1** and **NZ+1** for the third chamfer
- **BX+0**, **BY-1** and **BZ+0** as well as **NX-1**, **NY+0** and **NZ+1** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE VECTOR BX+1 BY+0 BZ+0 NX+0 NY-1 NZ+1 TURN MB MAX FMAX SYM-
   TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE VECTOR	Syntax initiator for the working plane definition by means of two vectors
BX, **BY** and **BZ**	Components of base vector, referenced to the workpiece coordinate system **W-CS**, for orienting the tilted X axis Input: **-99.9999999...+99.9999999**
NX, **NY** and **NZ**	Components of the normal vector, referenced to the **W-CS**, for orienting the tilted Z axis Input: **-99.9999999...+99.9999999**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1091 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1095 Optional syntax element

Notes

- If the components of the normal vector contain very small values, such as 0 or 0.0000001, the control cannot determine the working plane slope. In such cases, the control cancels machining with an error message. This behavior cannot be configured.
- The control calculates standardized vectors from the values you enter.

Notes about non-perpendicular vectors

To ensure that the definition of the working plane is unambiguous, the vectors must be programmed perpendicular to each other.

The machine manufacturer uses the optional machine parameter **autoCorrectVector** (no. 201207) to define the behavior of the control with non-perpendicular vectors.

As an alternative to an error message, the control can either correct or replace the non-perpendicular base vector. This correction (or replacement) does not affect the normal vector.

The correction behavior of the control if the base vector is not perpendicular:

- The control projects the base vector along the normal vector onto the working plane defined by the normal vector.

Correction behavior of the control if the base vector is not perpendicular and too short, parallel or antiparallel to the normal vector:

- If the normal vector contains the value 0 in the **NX** component, the base vector corresponds to the original X axis.
- If the normal vector contains the value 0 in the **NY** component, the base vector corresponds to the original Y axis.

Definition

Abbreviation	Definition
B (e.g., in **BX**)	Base vector
N (e.g., in **NX**)	Normal vector

PLANE POINTS

Application

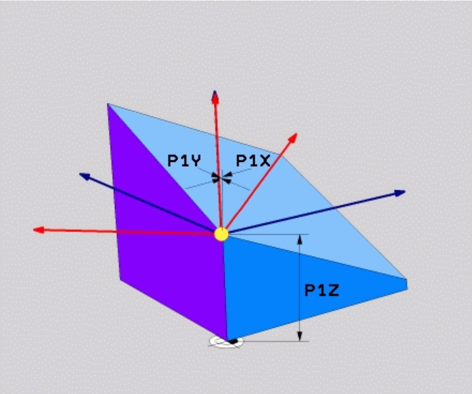
Use the **PLANE POINTS** function to define the working plane by three points.

Related topics

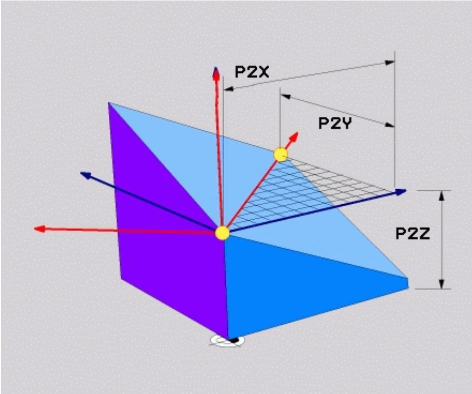
- Aligning the plane with touch probe cycle **431 MEASURE PLANE**
  Further information: "Cycle 431 MEASURE PLANE ", Page 1824

Description of function

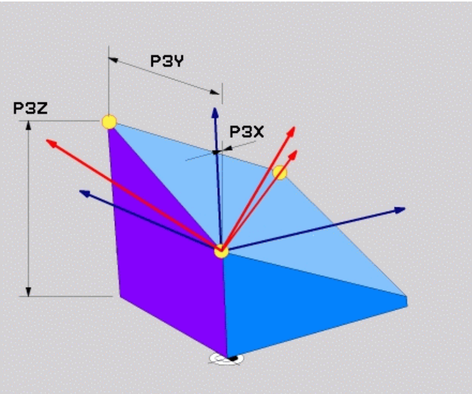
Points define a working plane by using their coordinates in the non-tilted workpiece coordinate system **W-CS**.



First point with coordinates **P1X**, **P1Y** and **P1Z**



Second point with coordinates **P2X**, **P2Y** and **P2Z**



Third point with coordinates **P3X**, **P3Y** and **P3Z**

All nine coordinates must be defined even if one or several coordinates equals 0.

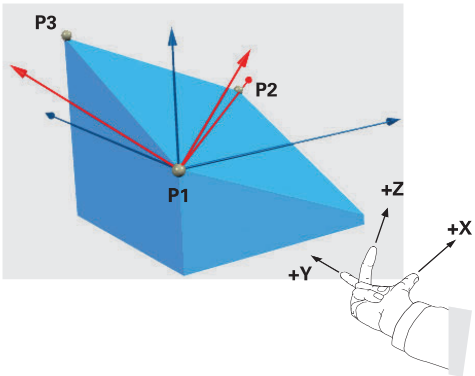
The first point with coordinates **P1X**, **P1Y** and **P1Z** defines the first point of the tilted X axis.

> You can imagine that the first point defines the origin of the tilted X axis and therefore the point serving for orientation of the working plane coordinate system **WPL-CS**.
>
> Ensure that the definition of the first point will not shift the workpiece datum. If the coordinates of the first point are to be programmed with the value 0, the workpiece datum may have to be shifted to that position before.

The second point with coordinates **P2X**, **P2Y** and **P2Z** defines the second point of the tilted X axis and consequently its orientation.

> The orientation of the tilted Y axis in the defined working plane results automatically because both axes are perpendicular to one another.

The third point with coordinates **P3X**, **P3Y** and **P3Z** defines the slope of the tilted working plane.



To direct the positive tool axis direction away from the workpiece, the following conditions apply to the position of the three points:

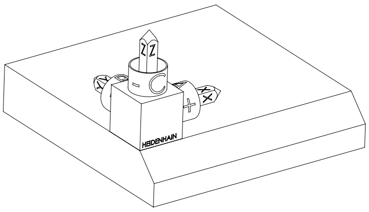
- Point 2 is to the right of point 1
- Point 3 is above the connecting lines between points 1 and 2

Application example

Example

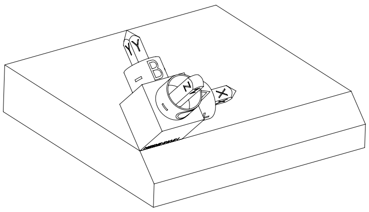
```
11 PLANE POINTS P1X+0  P1Y+0 P1Z+0 P2X+1 P2Y+0 P2Z+0 P3X+0 P3Y+1 P3Z+1
   TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis



Using the first two points **P1** and **P2**, the control orients the X axis of the **WPL-CS**.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

P3 defines the slope of the tilted working plane.

The orientations of the tilted Y and Z axes result automatically because all axes are perpendicular to one another.

> The drawing dimensions or any values which will not alter the ratio between the entered values can be used.
>
> In the example, **P2X** may also be defined by the workpiece width **+100**. **P3Y** and **P3Z** can also be programmed by using the chamfer width **+10**.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following points:
>
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X+0**, **P2Y+1**, **P2Z+0** and **P3X-1**, **P3Y+0**, **P3Z+1** for the second chamfer
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X-1**, **P2Y+0**, **P2Z+0** and **P3X+0**, **P3Y-1**, **P3Z+1** for the third chamfer
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X+0**, **P2Y-1**, **P2Z+0** and **P3X+1**, **P3Y+0**, **P3Z+1** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE POINTS P1X+0  P1Y+0 P1Z+0 P2X+1 P2Y+0 P2Z+0 P3X+0 P3Y+1 P3Z+1
   TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE POINTS	Syntax initiator for the working plane definition by means of three points
P1X, **P1Y** and **P1Z**	Coordinates of the first point of the tilted X axis, referenced to the workpiece coordinate system **W-CS** Input: **-999999999.999999...+999999999.999999**
P2X, **P2Y** and **P2Z**	Coordinates of the second point, referenced to the **W-CS** for orienting the tilted X axis Input: **-999999999.999999...+999999999.999999**
P3X, **P3Y** and **P3Z**	Coordinates of the third point, referenced to the **W-CS** for inclining the tilted working plane Input: **-999999999.999999...+999999999.999999**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1091 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1095 Optional syntax element

Definition

Abbreviation	Definition
P (e.g., in **P1X**)	Point

PLANE RELATIV

Application

Use the **PLANE RELATIV** function to define the working plane by just one spatial angle.

The defined angle always takes effect with reference to the input coordinate system **I-CS**.

Further information: "Reference systems", Page 1010

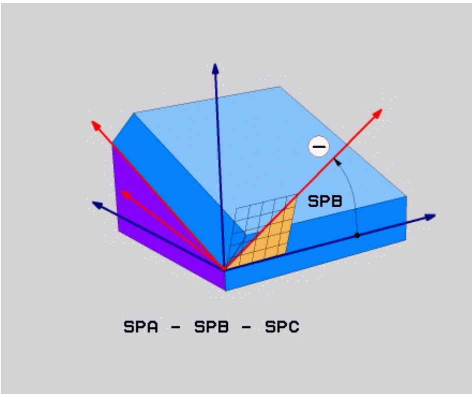
Description of function

A relative spatial angle defines a working plane as a rotation in the active reference system.

When the working plane is not tilted, the defined spatial angle is referenced to the non-tilted workpiece coordinate system **W-CS**.

When the working plane is tilted, the defined spatial angle is referenced to the working plane coordinate system **WPL-CS**.

> **PLANE RELATIV** allows, for example, programming a chamfer on a tilted workpiece surface by tilting the working plane further by the chamfer angle.



Additive spatial angle **SPB**

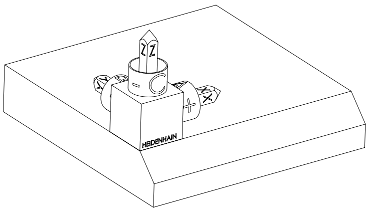
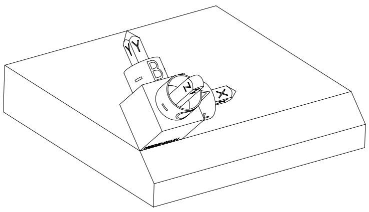
Each **PLANE RELATIV** function defines one spatial angle exclusively. However, it is possible to program any number of **PLANE RELATIV** functions in a row.

If you want to return the working plane that was active before the **PLANE RELATIV** function, define another **PLANE RELATIV** function with the same angle, but with the opposite algebraic sign.

Application example

Example

```
11 PLANE RELATIV SPA+45 TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the spatial angle **SPA+45**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **SPA** angle is around the non-tilted X axis. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following spatial angles:

- First PLANE RELATIVE function with **SPC+90** and another relative tilting with **SPA+45** for the second chamfer
- First PLANE RELATIVE function with **SPC+180** and another relative tilting with **SPA+45** for the third chamfer
- First PLANE RELATIVE function with **SPC+270** and another relative tilting with **SPA+45** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

When shifting the workpiece datum further in a tilted working plane, incremental values must be defined.

Further information: "Note", Page 1083

Input

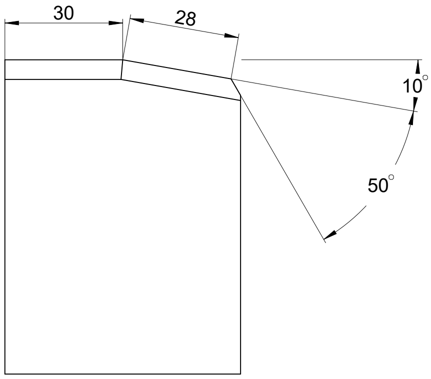
```
11 PLANE RELATIV SPA+45 TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE RELATIV	Syntax initiator for the working plane definition by means of one relative spatial angle
SPA, **SPB** or **SPC**	Rotation around the X, Y or Z axis of the workpiece coordinate system **W-CS** Input: **-360.0000000...+360.0000000** When the working plane is tilted, the rotation is in effect around the X, Y or Z axis in the working plane coordinate system **WPL-CS**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1091 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1095 Optional syntax element

Note

Incremental datum shift using a chamfer as example



50° chamfer on a tilted workpiece surface

Example

```
11 TRANS DATUM AXIS X+30
12 PLANE RELATIV SPB+10 TURN MB MAX FMAX SYM- TABLE ROT
13 TRANS DATUM AXIS IX+28
14 PLANE RELATIV SPB+50 TURN MB MAX FMAX SYM- TABLE ROT
```

This procedure offers the advantage of being able to program directly with the drawing dimensions.

Definition

Abbreviation	Definition
SP (e.g., in **SPA**)	Spatial

PLANE RESET

Application

Use the **PLANE RESET** function to reset all tilt angles and deactivate tilting of the working plane.

Description of function

The **PLANE RESET** function always executes two partial tasks:

- Reset all tilt angles, regardless of the selected tilt function or the type of angle
- Deactivate tilting of the working plane

> No other tilting function will carry out this partial task!
>
> Even when programming all angles with the value 0 in any tilting function, tilting of the working plane remains active.

The optional rotary axis positioning allows tilting the rotary axes back to the home position as the third partial task.

Further information: "Rotary axis positioning", Page 1088

Input

```
11 PLANE RESET TURN MB MAX FMAX
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE RESET	Syntax initiator for resetting all tilting angles and for deactivating an active tilting function
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088

Note

Before every program run, ensure that no undesired coordinate transformations are active. When needed, tilting of the working plane can also be deactivated manually in the **3-D rotation** window.

Further information: "3-D rotation window (option 8)", Page 1098

> The status display allows checking the desired status of the tilting situation.
>
> **Further information:** "Status display", Page 1056

PLANE AXIAL

Application

Use the **PLANE AXIAL** function to define the working plane with anywhere from one to three absolute or incremental axis angles.

An axis angle can be programmed for each rotary axis available on the machine.

> Because you are able to define just one axis angle, you can also use **PLANE AXIAL** on machines with just one rotary axis.

Please note that NC programs with axis angles always depend on the kinematics and therefore depend on the machine in question!

Related topics

- Programming independently of kinematics, using spatial angles

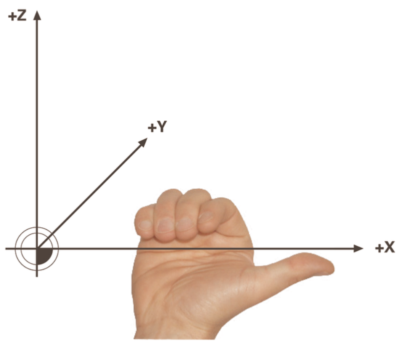
  Further information: "PLANE SPATIAL", Page 1059

Description of function

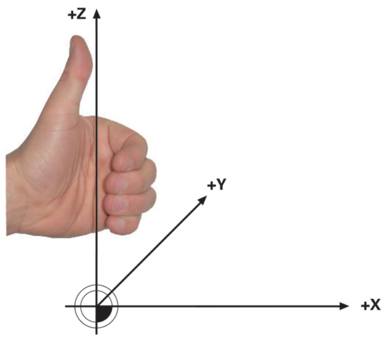
Axis angles define both the orientation of the working plane as well as the nominal coordinates of the rotary axes.

The axis angles must correspond to the axes present on the machine. If you try to program axis angles for rotary axes that do not exist on the machine, the control will generate an error message.

As the axis angles depend on the kinematics, a distinction must be made between the head and the table axes as far as the algebraic signs are concerned.



Extended right-hand rule for head rotary axes



Extended left-hand rule for table rotary axes

The thumb of the hand in question points in the positive direction of the axis around which the rotation occurs. If you curl your fingers, the curled fingers point in the positive direction of rotation.

Bear in mind that when working with rotary axes layered on top of one another, the positioning of the first rotary axis will also modify the position of the second rotary axis.

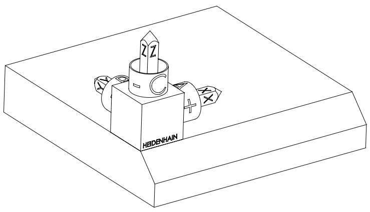
Application example

The example below applies to a machine with AC table kinematics whose two rotary axes are perpendicular and layered on top of one another.

Example

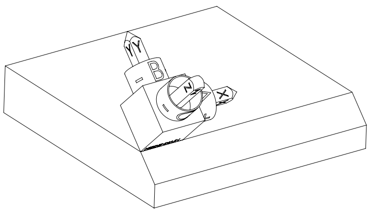
```
11 PLANE AXIAL A+45 TURN MB MAX FMAX
```

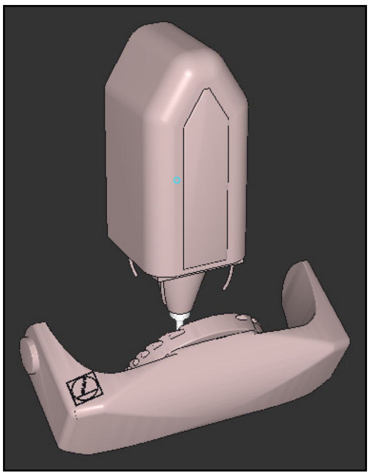
Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis





Using the defined axis angle **A**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by angle **A** is around the non-tilted X axis.

> To position the tool perpendicular to the chamfer surface, table rotary axis A must tilt to the rear.
>
> In accordance with the extended left-hand rule for table axes, the algebraic sign of the A axis value must be positive.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following axis angles:
>
> - **A+45** and **C+90** for the second chamfer
> - **A+45** and **C+180** for the third chamfer
> - **A+45** and **C+270** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE AXIAL A+45 TURN MB MAX FMAX
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE AXIAL	Syntax initiator for the working plane definition using one to three axis angle
A	When an A axis is available, nominal position of the A rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
B	When a B axis is available, nominal position of the B rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
C	When a C axis is available, nominal position of the C rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1088

The **SYM** or **SEQ** entries as well as **COORD ROT** or **TABLE ROT** are possible, but are not effective in conjunction with **PLANE AXIAL**.

Notes

Refer to your machine manual.

If your machine allows spatial angle definitions, you can continue your programming with **PLANE RELATIV** after **PLANE AXIAL**.

- The axis angles of the **PLANE AXIAL** function are modally effective. If you program an incremental axis angle, the control will add this value to the currently effective axis angle. If you program two different rotary axes in two successive **PLANE AXIAL** functions, the new working plane is derived from the two defined axis angles.
- The **PLANE AXIAL** function does not take basic rotation into account.
- When used in conjunction with **PLANE AXIAL**, the programmed transformations mirroring, rotation and scaling do not affect the position of the rotation point nor the orientation of the rotary axes.

  Further information: "Transformations in the workpiece coordinate system (W-CS)", Page 1016
- Without the use of a CAM system, **PLANE AXIAL** is convenient only with rotary axes positioned at right angles.

Rotary axis positioning

Application

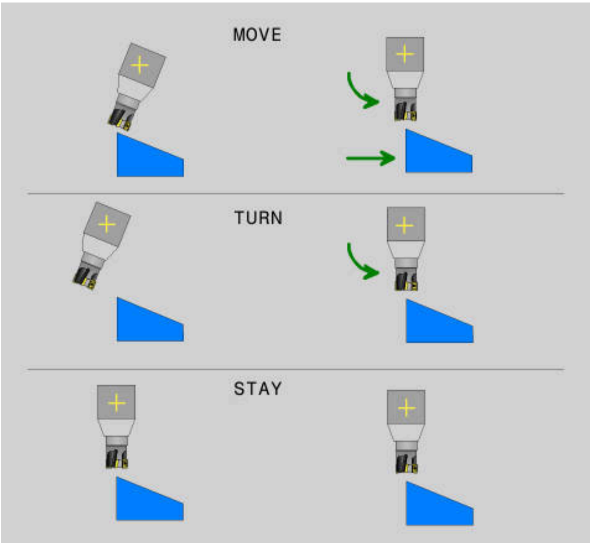
The type of rotary axis positioning defines how the control tilts the rotary axes to the calculated axis values.

The selection depends in part on the aspects below:

- Is the tool near the workpiece during tilting to position?
- Is the tool at a safe tilting position during tilting to position?
- May and can the rotary axes be positioned automatically?

Description of function

The control offers three types of rotary axis positioning from which one must be selected.



Type of rotary axis positioning	Meaning
MOVE	If you perform tilting near the workpiece, then use this option. **Further information:** "Rotary axis positioning MOVE", Page 1089
TURN	If the workpiece is so large that the range of traverse is not sufficient for the compensating movement of the linear axes, then use this option. **Further information:** "Rotary axis positioning TURN", Page 1089
STAY	The control does not position any axes. **Further information:** "Rotary axis positioning STAY", Page 1090

Rotary axis positioning MOVE

The control positions the rotary axes and performs compensation movements in the linear principal axes.

The compensation movements ensure that the relative position between the tool and the workpiece will not change during the positioning process.

NOTICE

Danger of collision!

The center of rotation is in the tool axis. In the case of large tool diameters, the tool may plunge into the material during tilting. During the tilting movement, there is a risk of collision!

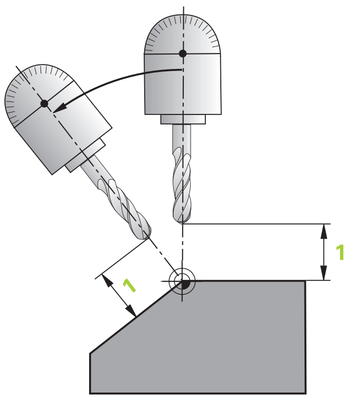
- Ensure sufficient distance between the tool and the workpiece

When **DIST** is not defined or when defining the value 0, the center of rotation and consequently the center of the compensation movements is in the tool tip.

When defining **DIST** with a value greater than 0, the center of rotation in the tool axis is shifted away from the tool tip by this value.

When wishing to tilt around a certain point on the workpiece, ensure the following:

- Prior to tilting to position, the tool is positioned directly above the desired point on the workpiece.
- The value defined in **DIST** matches exactly the clearance between the tool tip and the desired center of rotation.



Rotary axis positioning TURN

The control positions only the rotary axes. The tool must be positioned after tilting to position.

Rotary axis positioning STAY

Both the rotary axes and the tool must be positioned after tilting to position.

> Even with **STAY**, the control orients the working plane coordinate system **WPL-CS** automatically.

When selecting **STAY**, the rotary axes must be tilted to position in a separate positioning block after the **PLANE** function.

In the positioning block, use only the axis angles calculated by the control:

- **Q120** for the axis angle of the A axis
- **Q121** for the axis angle of the B axis
- **Q122** for the axis angle of the C axis

The variable avoids entry and calculating errors. In addition, no changes are required after changing the values within the **PLANE** functions.

Example

```
11 L A+Q120 C+Q122 FMAX
```

Input

MOVE

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 MOVE DIST0 FMAX
```

Selecting **MOVE** allows defining the syntax elements below:

Syntax element	Meaning
DIST	Distance between center of rotation and the tool tip Input: **0...99999999.9999999** Optional syntax element
F, **F AUTO** or **FMAX**	Feed rate definition for automatic rotary axis positioning Optional syntax element

TURN

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX
```

Selecting **TURN** allows defining the syntax elements below:

Syntax element	Meaning
MB	Retraction in the current tool axis direction before positioning the rotary axis Values with an incremental effect can be entered or a retraction up to the traverse limit can be defined by selecting **MAX**. Input: **0...99999999.9999999** or **MAX** Optional syntax element
F, **F AUTO** or **FMAX**	Feed rate definition for automatic rotary axis positioning Optional syntax element

STAY

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX
```

Selecting **STAY** does not allow defining further syntax elements.

Note

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect or no pre-positioning before tilting the tool into position can lead to a risk of collision during the tilting movement!

- Program a safe position before the tilting movement
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Tilting solution

Application

SYM (SEQ) allows selecting the desired option from several tilting solutions.

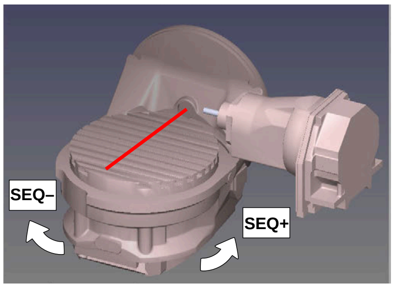
Unambiguous tilting solutions can be defined by using axis angles exclusively.

All other definition options can result in several tilting solutions, depending on the machine.

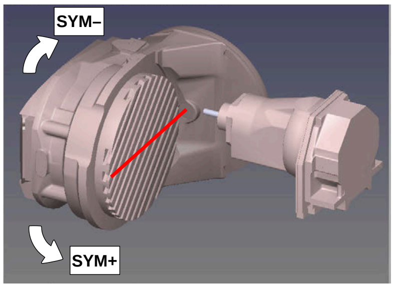
Description of function

The control offers two options from which one must be selected.

Option	Meaning
SYM	With **SYM** you select a tilting solution relative to the symmetry point of the master axis. **Further information:** "Tilting solution SYM", Page 1092
SEQ	With **SEQ** you select a tilting solution relative to the basic position of the master axis. **Further information:** "Tilting solution SEQ", Page 1093



Reference for **SEQ**



Reference for **SYM**

If the solution you have selected with **SYM** (**SEQ**) is not within the machine's range of traverse, then the control displays the **Entered angle not permitted** error message.

The entry of **SYM** or **SEQ** is optional.

If you do not define **SYM** (**SEQ**), then the control determines the solution as follows:

1. Check whether both possible solutions are within the traverse range of the rotary axes
2. Two possible solutions: Based on the current position of the rotary axes, choose the possible solution with the shortest path
3. One possible solution: Choose the only solution
4. No possible solution: Issue the error message **Entered angle not permitted**

Tilting solution SYM

With the **SYM** function, you select one of the possible solutions relative to the symmetry point of the master axis:

- **SYM+** positions the master axis in the positive half-space relative to the symmetry point
- **SYM-** positions the master axis in the negative half-space relative to the symmetry point

As opposed to **SEQ**, **SYM** uses the symmetry point of the master axis as its reference. Every master axis has two symmetry positions, which are 180° apart from each other (sometimes only one symmetry position is in the traverse range).

To determine the symmetry point:

- Perform **PLANE SPATIAL** with any spatial angle and **SYM+**
- Save the axis angle of the master axis in a Q parameter (e.g., –80)
- Repeat the **PLANE SPATIAL** function with **SYM-**
- Save the axis angle of the master axis in a Q parameter (e.g., –100)
- Calculate the average value (e.g., –90)

The average value corresponds to the symmetry point.

Tilting solution SEQ

With the **SEQ** function, you select one of the possible solutions relative to the home position of the master axis:

- **SEQ+** positions the master axis in the positive tilting range relative to the home position
- **SEQ-** positions the master axis in the negative tilting range relative to the home position

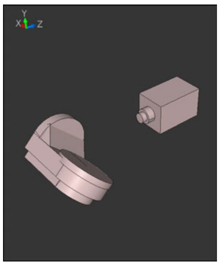
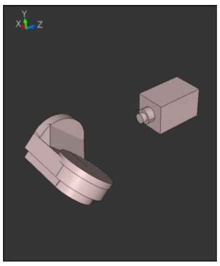
SEQ assumes that the master axis is in its home position (0°). Relative to the tool, the master axis is the first rotary axis, or the last rotary axis relative to the table (depending on the machine configuration). If both possible solutions are in the positive or negative range, then the control automatically uses the closer solution (shorter path). If you need the second possible solution, then you must either pre-position the master axis (in the area of the second possible solution) before tilting the working plane, or work with **SYM**.

Examples

Machine with C rotary axis and A tilting table.
Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

Limit switch	Start position	SYM = SEQ	Resulting axis position
None	A+0, C+0	Not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A–45, C–90
None	A+0, C–105	Not prog.	A–45, C–90
None	A+0, C–105	+	A+45, C+90
None	A+0, C–105	–	A–45, C–90
–90 < A < +10	A+0, C+0	Not prog.	A–45, C–90
–90 < A < +10	A+0, C+0	+	Error message
–90 < A < +10	A+0, C+0	–	A–45, C–90

Machine with B rotary axis and A tilting table (limit switches: A +180 and –100).
Programmed function: PLANE SPATIAL SPA-45 SPB+0 SPC+0

SYM	SEQ	Resulting axis position	Kinematics view
+		A–45, B+0	
-		Error message	**No solution in limited range**
	+	Error message	**No solution in limited range**
	-	A–45, B+0	

The position of the symmetry point is contingent on the kinematics. If you change the kinematics (such as switching the head), then the position of the symmetry point changes as well.

Depending on the kinematics, the positive direction of rotation of **SYM** may not correspond to the positive direction of rotation of **SEQ**. Therefore, ascertain the position of the symmetry point and the direction of rotation of **SYM** on each machine before programming.

Transformation types

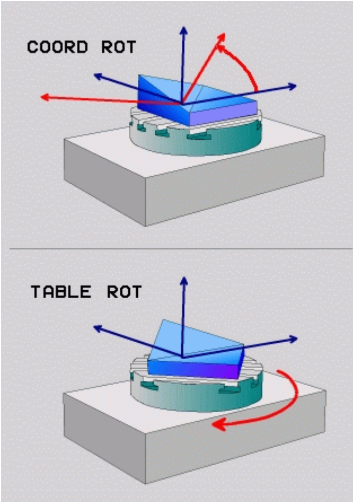
Application

COORD ROT and **TABLE ROT** influence the orientation of the working plane coordinate system **WPL-CS** through the axis position of a free rotary axis.

> Any rotary axis becomes a free rotary axis with the following configuration:
>
> - The rotary axis has no effect on the tool angle of inclination because the rotary axis and the tool axis are parallel in the tilting situation
> - The rotary axis is the first rotary axis in the kinematic chain starting from the workpiece
>
> The effect of the **COORD ROT** and **TABLE ROT** transformation types therefore depends on the programmed spatial angles and the machine kinematics.

Description of function

The control offers two options.



Option	Meaning
COORD ROT	> The control positions the free rotary axis to 0 > The control orients the working plane coordinate system in accordance with the programmed spatial angle
TABLE ROT	**TABLE ROT** with: ■ SPA **and** SPB **equal to** 0 ■ SPC **equal or unequal to** 0 > The control orients the free rotary axis in accordance with the programmed spatial angle > The control orients the working plane coordinate system in accordance with the basic coordinate system **TABLE ROT** with: ■ **At least** SPA **or** SPB **unequal to** 0 ■ SPC **equal or unequal to** 0 > The control does not position the free rotary axis. The position prior to tilting the working plane is maintained > Since the workpiece was not positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle

If no free rotary axis arises in a tilting situation, then the **COORD ROT** and **TABLE ROT** transformation types have no effect.

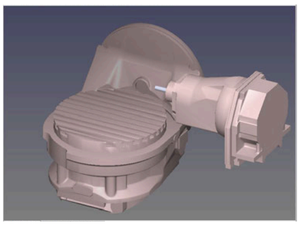
The entry of **COORD ROT** or **TABLE ROT** is optional.

If no transformation type was selected, then the control uses the **COORD ROT** transformation type for the **PLANE** functions

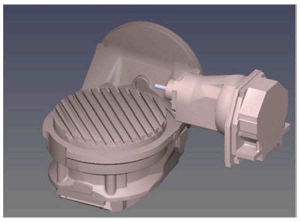
Example

The following example shows the effect of the **TABLE ROT** transformation type in conjunction with a free rotary axis.

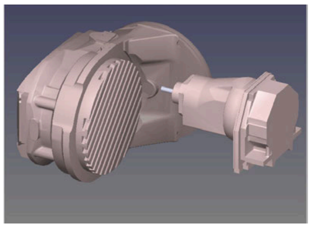
11 L B+45 R0 FMAX	; Pre-position the rotary axis
12 PLANE SPATIAL SPA-90 SPB+20 SPC +0 TURN F5000 TABLE ROT	; Tilt the working plane



Origin



A = 0, B = 45



A = −90, B = 45

- The control positions the B axis to the axis angle B+45
- With the programmed tilting situation with SPA−90, the B axis becomes the free rotary axis
- The control does not position the free rotary axis. The position of the B axis prior to the tilting of the working plane is maintained
- Since the workpiece was not also positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle SPB +20

Notes

- For the positioning behavior with the **COORD ROT** and **TABLE ROT** transformation types, it makes no difference whether the free rotary axis is a table axis or a head axis.
- The resulting axis position of the free rotary axis depends on an active basic rotation, among other factors.
- The orientation of the working plane coordinate system is also dependent on a programmed rotation (e.g., with Cycle **10 ROTATION**).

16.7.3 3-D rotation window (option 8)

Application

The **3-D rotation** window allows activating and deactivating tilting of the working plane for the **Manual** and **Program Run** operating modes. This allows restoring the tilted working plane and retracting the tool (e.g., after program cancellation in the **Manual operation** application).

Related topics

- Tilting the working plane in the NC program

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 1054
- Reference systems of the control

  Further information: "Reference systems", Page 1010

Requirements

- Machine with rotary axes
- Kinematics description

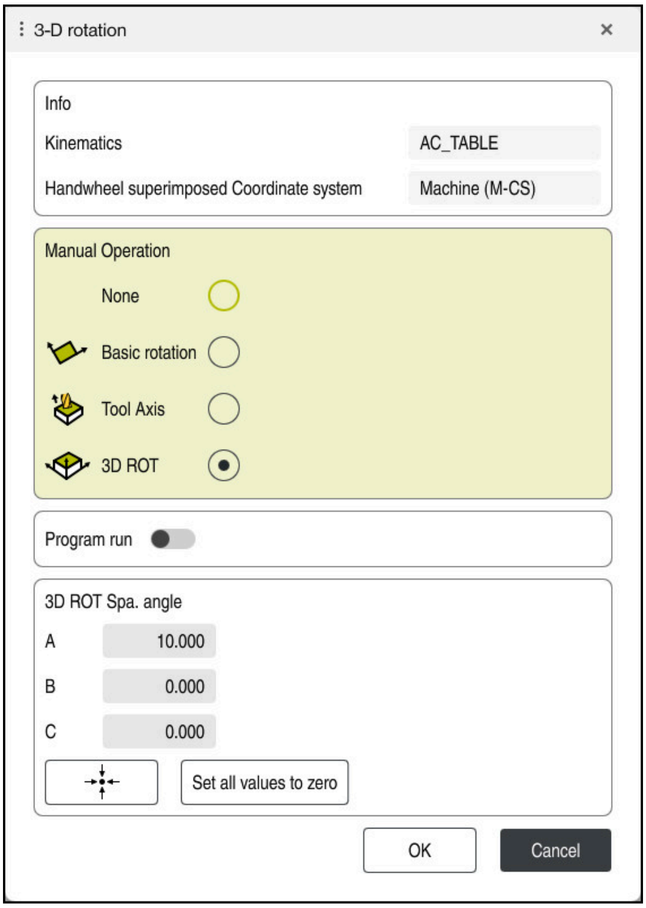
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 1 (software option 8)
- Function enabled by the machine manufacturer

  In the machine parameter **rotateWorkPlane** (no. 201201), the machine manufacturer defines whether tilting the working plane is allowed on the machine.
- Tool with tool axis **Z**

Description of function

The **3-D rotation** window can be opened with the **3D ROT** button in the **Manual operation** application.

Further information: "Manual operation application", Page 202



3-D rotation window

The **3-D rotation** window contains the following information:

Area	Contents
Info	Information about the machine: ■ Name of the active machine kinematics ■ Coordinate system in which handwheel superimpositioning is active **Further information:** "Reference systems", Page 1010 **Further information:** "Function Handwheel superimp.", Page 1225 **Further information:** "Activating handwheel superimpositioning with M118", Page 1334

Area	Contents
Manual Operation	Effect of the tilting function in the **Manual** operating mode: ■ **None** The control will not take the rotary axes positions that are not equal to 0 into account. Traverses take place in the **W-CS** workpiece coordinate system. **Further information:** "Workpiece coordinate system W-CS", Page 1016 ■ **Basic rotation** The control takes the columns **SPA**, **SPB** and **SPC** into account, but no rotary axis positions that are not equal to 0. Traverses take place in the **W-CS** workpiece coordinate system. **Further information:** "Basic rotation selection item", Page 1100 ■ **Tool axis** This is relevant only for head rotary axes. The traverses take place in the **T-CS** tool coordinate system. **Further information:** "Tool axis selection item", Page 1101 ■ **3D ROT** The control takes the positions of rotary axes and columns **SPA**, **SPB** and **SPC** of the preset table into account. The traverses take place in the **WPL-CS** working plane coordinate system. **Further information:** "3D ROT selection item", Page 1101
Program run	When activating the **Tilt working plane** function for the **Program run** operating mode, the entered angle of rotation applies starting from the first NC block of the NC program to be run. If you use Cycle **19 WORKING PLANE** or the **PLANE** function in the NC program, then the angular values defined there become active. The control will reset the entered angular values to 0.
3D ROT Spa. angle	Currently active angle for the **3D ROT** selection item The machine manufacturer uses the machine parameter **planeOrientation** (no. 201202) to define whether the control calculates with spatial angles **SPA**, **SPB** and **SPC** or with the axis values of the existing rotary axes.

Confirm the selection with **OK**. If a selection item is active in the **Manual Operation** or **Program run** areas, then the control highlights the area in green.

If a selection item is active in the **3-D rotation** window, then the control displays the appropriate symbol in the **Positions** workspace.

Further information: "Positions workspace", Page 163

Basic rotation selection item

If you select the **Basic rotation** selection item, then the axes move, taking into account a basic rotation or a 3D basic rotation.

Further information: "Basic rotation and 3D basic rotation", Page 1027

The axis movements take effect in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 1016

If the active workpiece preset contains a basic rotation or 3D basic rotation, the control additionally displays the corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 163

The **3D ROT Spa. angle** area has no function with this selection item.

Tool axis selection item

If you select the **Tool axis** selection item, then you can move in the positive or negative direction of the tool axis. The control locks all other axes. This selection item makes sense only for machines with rotary head axes.

The traverse movement is active in the **T-CS** tool coordinate system.

Further information: "Tool coordinate system T-CS", Page 1022

This selection item can be used, for example, in the following cases:

- When retracting the tool in the direction of the tool axis during an interruption of a 5-axis machining program.
- When traversing with the axis keys or the handwheel with a pre-positioned tool.

The **3D ROT Spa. angle** area has no function with this selection item.

3D ROT selection item

If you select the **3D ROT** selection item, then all axes move in the tilted machining plane. The traversing movements are active in the **WPL-CS** working plane coordinate system.

Further information: "Working plane coordinate system WPL-CS", Page 1018

If a basic rotation or 3D basic rotation has additionally been saved to the preset table, then it will automatically be taken into account.

In the **3D ROT Spa. angle** area, the control shows the currently active angle. The spatial angle can also be edited.

> If you edit the values in the **3D ROT Spa. angle** area, then you must position the rotary axes (e.g., in the **MDI** application).

Notes

- The control uses the **COORD ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **COORD ROT**
  - after **PLANE RESET**
  - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer
- The control uses the **TABLE ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **TABLE ROT**
  - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (option 8). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.

  In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- A tilted working plane will remain active even after a control restart.

  Further information: "Referencing workspace", Page 197
- PLC positionings defined by the machine manufacturer are not allowed when the working plane is tilted.

16.8 Inclined machining (option 9)

Application

When pre-positioning the tool during machining, workpiece positions that are difficult to reach can be machined without collisions.

Related topics

- Compensating for the tool angle of inclination with **FUNCTION TCPM** (option 9)

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
- Compensating for the tool angle of inclination with **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1341
- Tilting the working plane (option 8)

  Further information: "Tilting the working plane (option 8)", Page 1053
- Presets on the tool

  Further information: "Presets on the tool", Page 271
- Reference systems

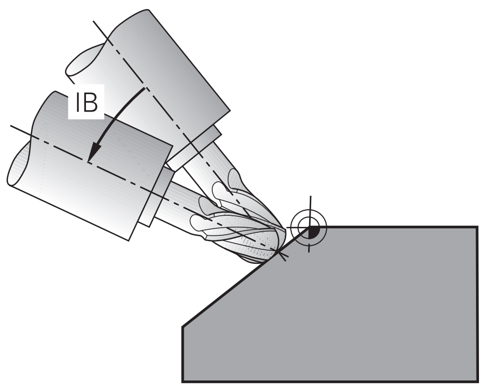
  Further information: "Reference systems", Page 1010

Requirements

- Machine with rotary axes
- Kinematics description

  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 2 (software option 9)

Description of function



The **FUNCTION TCPM** function allows executing inclined machining. In this process, one working plane may be tilted.

Further information: "Tilting the working plane (option 8)", Page 1053

Inclined machining can be implemented using the following functions:

- Incremental traverse of rotary axis

  Further information: "Inclined machining with incremental process", Page 1103
- Normal vectors

  Further information: "Inclined machining using normal vectors", Page 1103

Inclined machining with incremental process

Inclined machining can be implemented by changing the inclination angle in addition to the normal linear movement while function **FUNCTION TCPM** or **M128** is active, e. g. **L X100 Y100 IB-17 F1000 G01 G91 X100 Y100 IB-17 F1000**. In this process, the relative position of the tool's center of rotation remains the same while inclining the tool.

Example

* - ...	
12 L Z+50 R0 FMAX	; Position at clearance height
13 PLANE SPATIAL SPA+0 SPB-45 SPC +0 MOVE DIST50 F1000	; Define and activate the PLANE function
14 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS	; Activate TCPM
15 L IB-17 F1000	; Pre-position the tool
* - ...	

Inclined machining using normal vectors

In case of inclined machining using normal vectors, the tool angle of inclination is achieved by means of straight lines **LN**.

To execute inclined machining with normal vectors, function **FUNCTION TCPM** or miscellaneous function **M128** must be activated.

Example

* - ...	
12 L Z+50 R0 FMAX	; Position at clearance height
13 PLANE SPATIAL SPA+0 SPB+45 SPC +0 MOVE DIST50 F1000	; Tilt the working plane
14 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS	; Activate TCPM
15 LN X+31.737 Y+21,954 Z+33,165 NX+0,3 NY+0 NZ+0,9539 F1000 M3	; Incline the tool with the normal vector
* - ...	

16.9 Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)

Application

The **FUNCTION TCPM** function allows you to influence the positioning behavior of the control. When activating **FUNCTION TCPM**, the control compensates for any changed tool angles of inclination by means of compensating movements of the linear axes.

FUNCTION TCPM allows, for example, changing the tool angle of inclination for inclined machining while the position of the tool location point relative to the contour remains the same.

> Instead of **M128**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM**.

Related topics

- Compensating for the tool angle of inclination with **M128**

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1341
- Tilting the working plane

  Further information: "Tilting the working plane (option 8)", Page 1053
- Presets on the tool

  Further information: "Presets on the tool", Page 271
- Reference systems

  Further information: "Reference systems", Page 1010

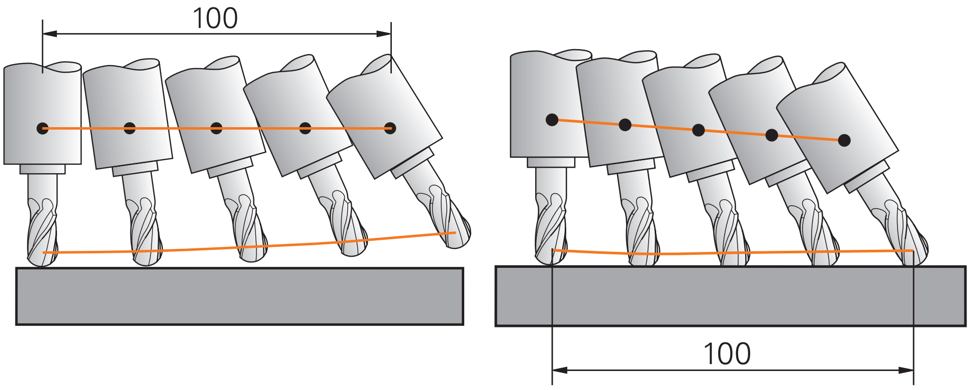
Requirements

- Machine with rotary axes
- Kinematics description

  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 2 (software option 9)

Description of function

FUNCTION TCPM is an improvement on the **M128** function which allows defining the behavior of the control while during the positioning of rotary axes.



Behavior without **TCPM**

Behavior with **TCPM**

When **FUNCTION TCPM** is active, the control shows the **TCPM** icon in the position display.

Further information: "Positions workspace", Page 163

The **FUNCTION RESET TCPM** function resets the **FUNCTION TCPM** function.

Input

FUNCTION TCPM

```
10 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT CENTER-CENTER F1000
```

The NC function contains the following syntax elements:

Syntax element	Meaning
FUNCTION TCPM	Syntax initiator for compensating tool angles of inclination
F TCP or **F CONT**	Interpretation of the programmed feed rate **Further information:** "Interpretation of the programmed feed rate ", Page 1106
AXIS POS or **AXIS SPAT**	Interpretation of programmed rotary axis coordinates **Further information:** "Interpretation of the programmed rotary axis coordinates", Page 1106
PATHC-TRL AXIS or **PATHCTRL VECTOR**	Interpolation of tool angle of inclination **Further information:** "Interpolation of tool angle of inclination between start and end positions", Page 1107
REFPNT TIP-TIP, **REFPNT TIP-CENTER** or **REFPNT CENTER-CENTER**	Selection of tool location point and tool rotation point **Further information:** "Selection of tool location point and tool rotation point", Page 1108 Optional syntax element
F	Maximum feed rate for compensating movements in the linear axes for movements with a rotary-axis component **Further information:** "Limiting the linear-axis feed rate", Page 1109 Optional syntax element

FUNCTION RESET TCPM

```
10 FUNCTION RESET TCPM
```

The NC function contains the following syntax elements:

Syntax element	Meaning
FUNCTION RESET TCPM	Syntax initiator for resetting of **FUNCTION TCPM**

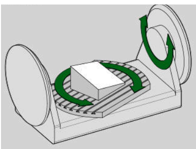
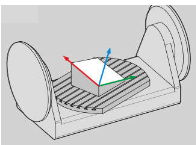
Interpretation of the programmed feed rate

The control offers the following options for interpreting the feed rate:

Selection	Function
F TCP	When selecting **F TCP**, the control interprets the programmed feed rate as the relative speed between the tool location point and the workpiece.
F CONT	When selecting **F CONT**, the control interprets the programmed feed rate as contouring feed rate. In this process, the control transfers the contouring feed rate to the respective axes of the active NC block.

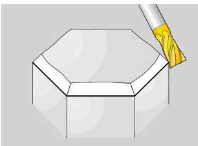
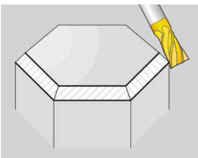
Interpretation of the programmed rotary axis coordinates

The control offers the options below for interpreting the tool angle of inclination between the start and end position:

Selection	Function
 AXIS POS	When selecting **AXIS POS**, the control interprets the programmed rotary axis coordinates as axis angle. The control positions the rotary axes on the position defined in the NC program. The **AXIS POS** selection is primarily suitable in conjunction with perpendicularly arranged rotary axes. **AXIS POS** can only be used with different machine kinematics (e.g., 45° swivel heads) if the programmed rotary axis coordinates define the desired working plane alignment correctly (e.g., using a CAM system).
 AXIS SPAT	If **AXIS SPAT** is selected, the control interprets the programmed rotary axis coordinates as spatial angles. The control preferably implements the spatial angles as orientation of the coordinate system and tilts only required axes. Select **AXIS SPAT** to allow using NC programs regardless of kinematics. The **AXIS SPAT** selection item defines the spatial angles relative to the **I-CS** input coordinate system. The defined angles have the effect of incremental spatial angles. In the first traversing block after the function **FUNCTION TCPM**, always program with **AXIS SPAT**, **SPA**, **SPB** and **SPC**, including with spatial angles of 0°. **Further information:** "Input coordinate system I-CS", Page 1021

Interpolation of tool angle of inclination between start and end positions

The control offers the options below for interpolating the tool angle of inclination between the programmed start and end positions:

Selection	Function
 **PATHCTRL AXIS**	When selecting **PATHCTRL AXIS**, the control interpolates linearly between the start and end point. Use **PATHCTRL AXIS** with NC programs with small changes of the tool angle of inclination per NC block. In this case, the angle **TA** in Cycle **32** can be large. **Further information:** "Cycle 32 TOLERANCE ", Page 1213 **PATHCTRL AXIS** can be used both for face milling and also for peripheral milling. **Further information:** "3D tool compensation during face milling (option 9)", Page 1130 **Further information:** "3D tool compensation during peripheral milling (option 9)", Page 1137
 **PATHCTRL VECTOR**	If **PATHCTRL VECTOR** is selected, the tool orientation within an NC block always lies in the plane that is defined by the start orientation and end orientation. With **PATHCTRL VECTOR** the control generate a plane surface even if there are large changes in the tool inclination angle. Use **PATHCTRL VECTOR** for peripheral milling if there are large changes in the tool inclination angle per NC block.

In both cases, the control moves the programmed tool location point on a straight line between the start position and end position.

To obtain continuous movement, define Cycle **32** with a **tolerance for rotary axes**.

Further information: "Cycle 32 TOLERANCE ", Page 1213

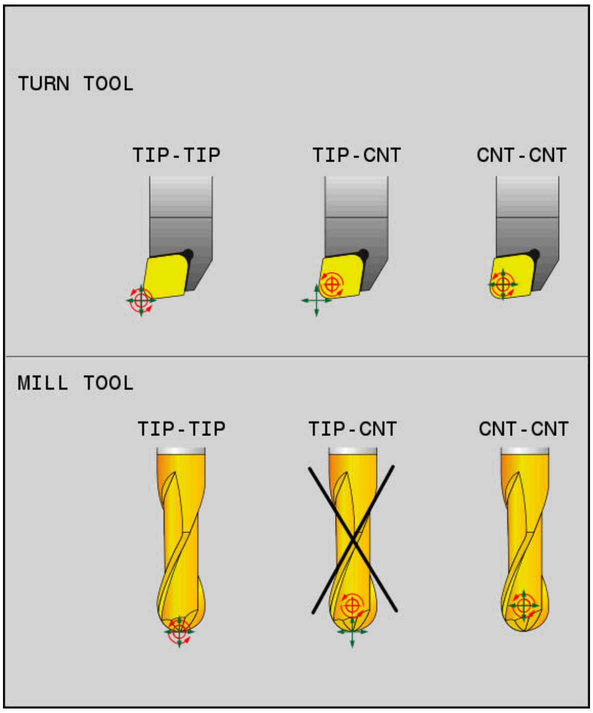
Selection of tool location point and tool rotation point

The control offers the options below for defining the tool location point and the tool rotation point:

Selection	Function
REFPNT TIP-TIP	When selecting **REFPNT TIP-TIP**, the tool location point and the tool rotation point are located at the tool tip.
REFPNT TIP-CENTER	When selecting **REFPNT TIP-CENTER**, the tool location point is located at the tool tip. The tool rotation point is located at the tool center point. The option **REFPNT TIP-CENTER** is optimized for turning tools (option 50). When the control positions the rotary axes, the tool rotation point remains at the same position. This allows you, for example, to machine complex contours by simultaneous turning. **Further information:** "Theoretical and virtual tool tip", Page 1118
REFPNT CENTER-CENTER	When selecting **REFPNT CENTER-CENTER**, the tool location point and the tool rotation point are located at the tool center point. Selecting **REFPNT CENTER-CENTER** allows executing CAM-generated NC programs which are referenced to the tool center point and still calibrate the tool relative to its tip. This allows the control to monitor the entire tool length for collisions while machining is in progress. Previously, this functionality could only be achieved by shortening the tool with **DL** and without the control monitoring the remaining tool length. **Further information:** "Tool data within variables", Page 1114 If you use **REFPNT CENTER-CENTER** to program pocket milling cycles, the control generates an error message. **Further information:** "Overview", Page 507

Further information: "Presets on the tool", Page 271

The reference point is optional. If you do not enter anything, the control uses **REFPNT TIP-TIP**.



Selection options of tool preset and tool rotation point

Limiting the linear-axis feed rate

The optional input of **F** allows you to limit the feed rate of linear axes for motions with a rotary-axis component.

Thus, you can avoid fast compensation movements (e.g., in case of retraction movement at rapid traverse).

> Make sure to select a value for the linear axis feed-rate limit that is not too small because large feed-rate variations may occur at the tool location point. Feed-rate variations impair the surface quality.
>
> If **FUNCTION TCPM** is active, the feed-rate limit affect only motions with a rotary-axis component, not for entirely linear motions.

The linear axis feed-rate limit remains in effect until you program a new value or reset **FUNCTION TCPM**.

Notes

NOTICE
Danger of collision!
Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation.
▶ Make sure to retract the tool before changing the position of the rotary axis

- Before positioning axes with **M91** or **M92,** and before a **TOOL CALL** block, reset the **FUNCTION TCPM** function.
- The following cycles can be used with active **FUNCTION TCPM**:
  - Cycle **32 TOLERANCE**
  - Cycle **800 ADJUST XZ SYSTEM** (option 50)
  - Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** (option 158)
  - Cycle **883 TURNING SIMULTANEOUS FINISHING** (option 158)
  - Cycle **444 PROBING IN 3-D**
- Use only ball-nose cutters for face milling in order to avoid contour damage. In combination with other tool shapes, check the NC program for any possible contour damage, using the **Simulation** workspace.

  Further information: "Notes", Page 1344

Notes about machine parameters

The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. For **FUNCTION TCPM** and **M128**, the machine parameter applies only to the rotary axis that rotates about the tool axis (in most cases **C_OFFS**).

Further information: "Basic transformation and offset", Page 2039

- If the machine parameter axis has not been defined or has been set to **TRUE**, the offset can be used to compensate a misalignment of the workpiece in the plane. The offset affects the orientation of the workpiece coordinate system **W-CS**.

  Further information: "Workpiece coordinate system W-CS", Page 1016
- If the machine parameter axis has been defined with **FALSE**, the offset cannot be used to compensate a misalignment of the workpiece in the plane. The control will not take the offset into account when executing the commands.

17

Compensations

17.1 Tool compensation for tool length and tool radius

Application

Delta values allow implementing tool compensation of the tool length and the tool radius. Delta values influence the calculated and therefore the active tool dimensions.

The tool length delta value **DL** is active in the tool axis. The tool radius delta value **DR** is active exclusively for radius-compensated traverses with the path functions and cycles.

Further information: "Path Functions", Page 317

Related topics

- Tool radius compensation

  Further information: "Tool radius compensation", Page 1114
- Tool compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 1120

Description of function

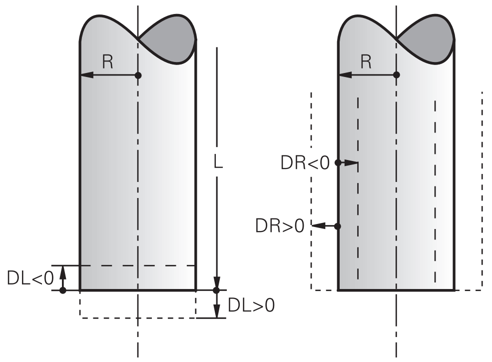
The control distinguishes between two types of delta values:

- Delta values within the tool table serve for permanent tool compensation that is required (e.g., due to wear).

  These delta values can be determined, for example, by using a tool touch probe. The control automatically enters the delta values in the tool management.

  Further information: "Tool management ", Page 297
- Delta values within a tool call serve for a tool compensation that is active exclusively in the current NC program (e.g., a workpiece oversize).

  Further information: "Tool call by TOOL CALL", Page 304



Delta values represent deviations from the length and radius of a tool.

A positive delta value enlarges the current tool length or the tool radius. The tool then cuts less material during machining (e.g., for a workpiece oversize).

A negative delta value reduces the current tool length or the tool radius. The tool then cuts more material during machining.

For programming delta values in an NC program, define the value within a tool call or by using a compensation table.

Further information: "Tool call by TOOL CALL", Page 304

Further information: "Tool compensation with compensation tables", Page 1120

Delta values within a tool call can also be defined by using variables.

Further information: "Tool data within variables", Page 1114

Tool length compensation

The control takes the tool length compensation into account as soon as a tool is called. The control performs tool length compensation only on tools of length L>0.

In tool length compensation, the control takes delta values from the tool table and the NC program into account.

Active tool length = **L** + $\mathbf{DL}_{\mathrm{TAB}}$ + $\mathbf{DL}_{\mathrm{Prog}}$

L:	Tool length **L** from the tool table **Further information:** "Tool table tool.t", Page 1995
$\mathbf{DL}_{\mathrm{TAB}}$:	Tool length delta value **DL** from the tool table **Further information:** "Tool table tool.t", Page 1995
$\mathbf{DL}_{\mathrm{Prog}}$:	Tool length delta value **DL** from the tool call or the compensation table The most recently programmed value becomes active. **Further information:** "Tool call by TOOL CALL", Page 304 **Further information:** "Tool compensation with compensation tables", Page 1120

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

Tool radius compensation

The control takes the tool radius compensation into account in the following cases:

- In case of active radius compensation **RR** or **RL**
  Further information: "Tool radius compensation", Page 1114
- Within machining cycles
  Further information: "Machining Cycles", Page 475
- For straight lines **LN** with surface normal vectors
  Further information: "Straight line LN", Page 1127

In tool radius compensation, the control takes the delta values from the tool table and the NC program into account.

Active tool radius = **R** + $\mathbf{DR}_{\mathrm{TAB}}$ + $\mathbf{DR}_{\mathrm{Prog}}$

R:	Tool radius **R** from the tool table **Further information:** "Tool table tool.t", Page 1995
$\mathbf{DR}_{\mathrm{TAB}}$:	Tool radius delta value **DR** from the tool table
$\mathbf{DR}_{\mathrm{Prog}}$:	Tool radius delta value **DR** from the tool call or the compensation table The most recently programmed value becomes active. **Further information:** "Tool call by TOOL CALL", Page 304 **Further information:** "Tool compensation with compensation tables", Page 1120

Tool data within variables

When executing a tool call, the control calculates all tool-specific values and saves them within variables.

Further information: "Preassigned Q parameters", Page 1368

Active tool length and tool radius:

Q parameters	Function
Q108	ACTIVE TOOL RADIUS
Q114	ACTIVE TOOL LENGTH

After the control has saved the current values within variables, the variables can be used in the NC program.

Application example

You can use the Q parameter **Q108 ACTIVE TOOL RADIUS** in order to shift the tool center point of the ball-nose cutter to the sphere center using the delta value for the tool length.

```
11 TOOL CALL "BALL_MILL_D4" Z S10000
12 TOOL CALL  DL-Q108
```

This allows the control to monitor the complete tool for collisions and the dimensions used in the NC program can still be programmed with reference to the ball center.

Notes

- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.

  Further information: "Simulation of tools", Page 1544
- The machine manufacturer uses the optional machine parameter **progToolCallDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

  Further information: "Tool call", Page 304

  Further information: "Positions workspace", Page 163
- The control takes up to six axes including the rotary axes into account in the tool compensation.

17.2 Tool radius compensation

Application

When tool radius compensation is active, the control will no longer reference the positions in the NC program to the tool center point, but to the cutting edge.

Use tool radius compensation to program drawing dimensions without having to consider the tool radius. This lets you use a tool with deviating dimensions without having to modify the program after a tool has broken.

Related topics

- Presets on the tool

  Further information: "Presets on the tool", Page 271

Requirements

- Defined tool data in the tool management

  Further information: "Tool management ", Page 297

Description of function

The control takes the active tool radius into account during tool radius compensation. The active tool radius results from the tool radius R and the delta values **DR** from the tool management and the **NC program**.

Active tool radius = **R** + $\mathbf{DR}_{TAB}$ + $\mathbf{DR}_{Prog}$

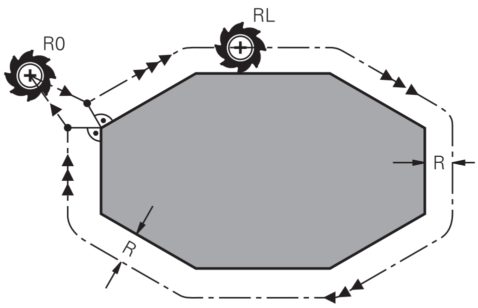
Further information: "Tool compensation for tool length and tool radius", Page 1112

Paraxial traverses can be compensated as follows:

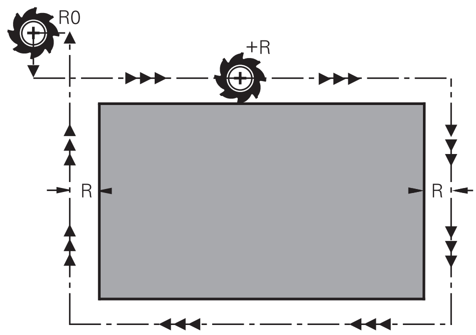
- **R+**: lengthens a paraxial traverse by the amount of the tool radius
- **R-**: shortens a paraxial traverse by the amount of the tool radius

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

- **RL**: tool radius compensation, on the left of the contour
- **RR**: tool radius compensation, on the right of the contour
- **R0**: resets an active tool radius compensation, positioning with the tool center point

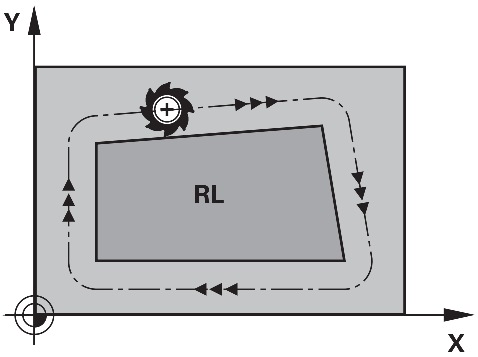


Radius-compensated traverse with path functions

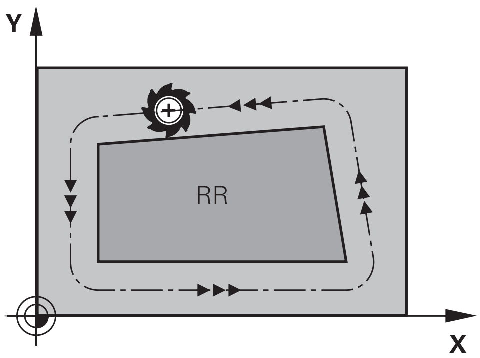


Radius-compensated traverse with paraxial movements

The tool center moves along the contour at a distance equal to the radius. **Right** or **left** are to be understood as based on the direction of tool movement along the workpiece contour.



RL: The tool moves on the left of the contour



RR: The tool moves on the right of the contour

Effect

Tool radius compensation is active starting from the NC block in which tool radius compensation is programmed. Tool radius compensation is effective modally and at the end of the block.

> Program tool radius compensation only once, allowing for quicker implemention of changes, for example.

The control resets tool radius compensation in the following cases:

- Positioning block with **R0**
- **DEP** function for departing from the contour
- Selection of a new NC program

Notes

NOTICE

Danger of collision!

The control needs safe positions for contour approach and departure. These positions must enable the control to perform compensating movements when radius compensation is activated and deactivated. Incorrect positions can lead to contour damage. Danger of collision during machining!

- Program safe approach and departure positions at a sufficient distance from the contour
- Consider the tool radius
- Consider the approach strategy

- When tool radius compensation is active, the control displays an icon in the **Positions** workspace.

  Further information: "Positions workspace", Page 163
- Between two NC blocks, each with a different tool radius compensation **RR** and **RL**, there must be at least one traversing block in the working plane without tool radius compensation **R0**.
- The control takes up to six axes including the rotary axes into account in the tool compensation.

Notes in connection with the machining of corners

- Outside corners:
  If you program radius compensation, the control moves the tool around outside corners on a transitional arc. If necessary, the control reduces the feed rate at outside corners during, for example, large changes in direction
- Inside corners:
  The control calculates the intersection of the tool center paths at inside corners under radius compensation. Starting at this point, the tool moves along the next contour element. This prevents damage to the workpiece at the inside corners. As a result, the tool radius for a certain contour cannot be selected to be just any size.

17.3 Tooth radius compensation for turning tools (option 50)

Application

The tip of a lathe tool has a certain radius (**RS**). During the machining of tapers, chamfers and radii, this results in distortions on the contour because the programmed traverse paths are referenced to the theoretical tool tip S. Tooth radius compensation (TRC) prevents the resulting deviations.

Related topics

- Tool data of turning tools

  Further information: "Tool data", Page 275
- Radius compensation with **RR** and **RL** in milling mode

  Further information: "Tool radius compensation", Page 1114

Requirement

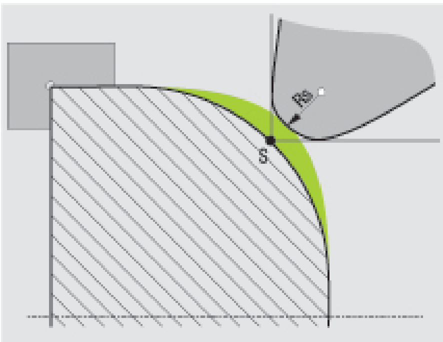
- Combined milling/turning (software option 50)
- Required tool data defined for the tool type

  Further information: "Tool data for the tool types", Page 284

Description of function

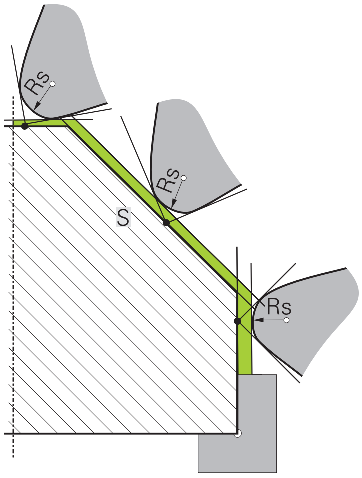
The control checks the cutting geometry with the point angle **P-ANGLE** and the setting angle **T-ANGLE**. Contour elements in the cycle are processed by the control only as far as this is possible with the specific tool.

In the turning cycles, the control automatically carries out tool radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **RL** or **RR**.



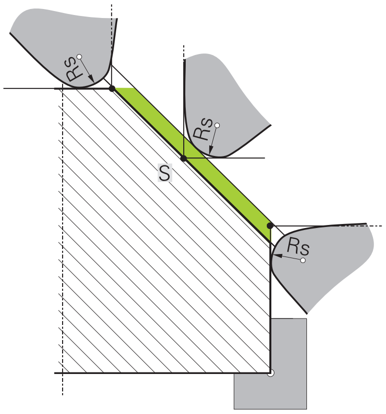
Offset between the tooth radius **RS** and the theoretical tool tip S.

Theoretical and virtual tool tip



Inclined surface with theoretical tool tip

The theoretical tool tip is active in the tool coordinate system. When the tool is inclined, the position of the tool tip rotates with the tool.



Inclined surface with virtual tool tip

To activate the virtual tool tip, use **FUNCTION TCPM** with the **REFPNT TIP-CENTER** selection item. Correct tool data are required for calculating the virtual tool tip.

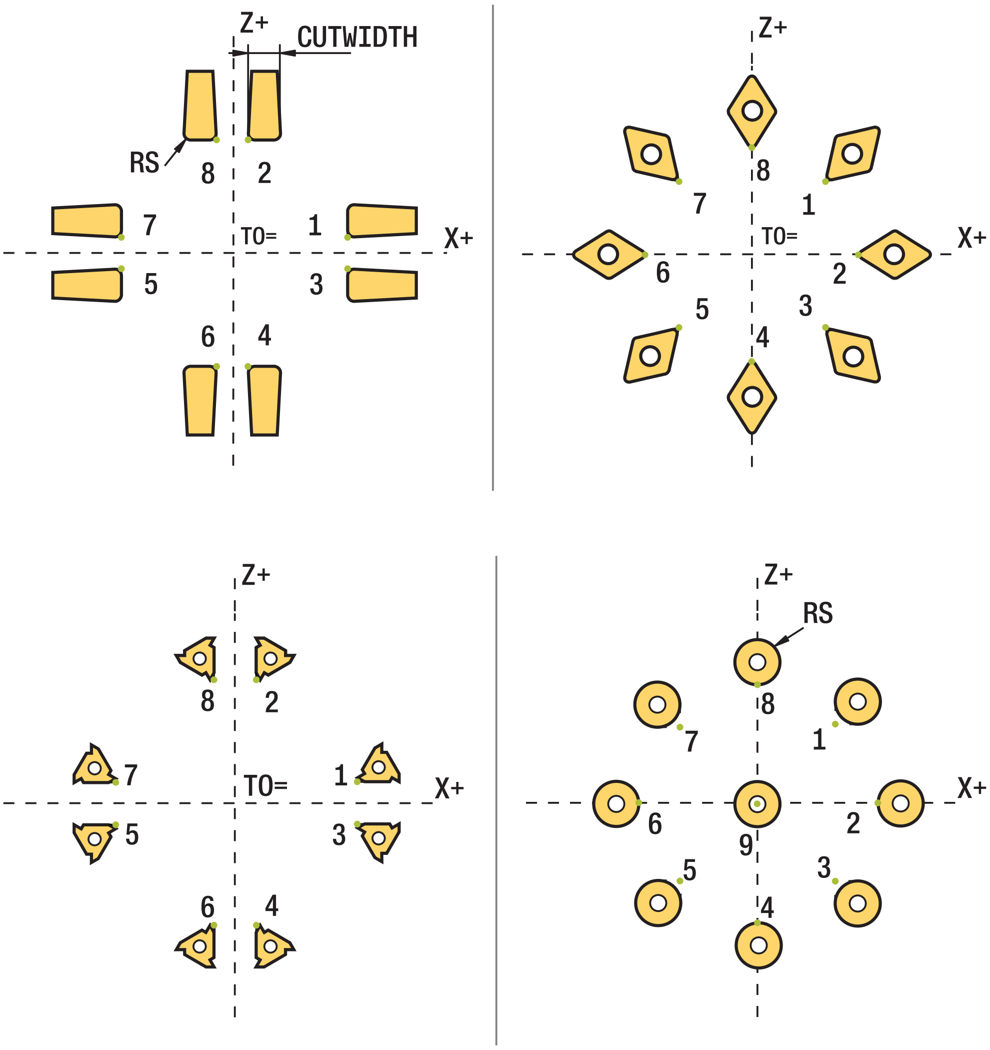
Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

The virtual tool tip is active in the workpiece coordinate system. When the tool is inclined, the virtual tool tip remains unchanged as long as the tool orientation **TO** is the same. The control automatically switches the status display **TO** and thus also the virtual tool tip if the tool leaves the angle range valid for **TO 1**, for example.

The virtual tool tip enables you to perform inclined paraxial longitudinal and transverse machining operations with high contour accuracy even without radius compensation.

Further information: "Simultaneous turning", Page 242

Notes



- The direction of the radius compensation is not clear when the tool-tip position (**TO=2, 4, 6, 8**) is neutral. In this case, TRC is only possible within fixed machining cycles.
- Tooth radius compensation is also possible during inclined machining.

  Active miscellaneous functions limit the possibilities here:
  - With **M128** tool-tip radius compensation is possible only in combination with machining cycles
  - **M144** or **FUNCTION TCPM** with **REFPNT TIP-CENTER** also allows tooth radius compensation with all positioning blocks, e.g. with **RL**/**RR**
- The control displays a warning when residual material is left behind due to the angle of the secondary cutting edges. You can suppress this warning with the machine parameter **suppressResMatlWar** (no. 201010).

17.4 Tool compensation with compensation tables

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 saved compensations can be called during the NC program for compensating the tool.

The compensation tables offer the following benefits:

- Values can be changed without adapting the NC program
- Values can be changed during NC program run

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**)

Further information: "Reference systems", Page 1010

Related topics

- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 2055

  Further information: "Compensation table *.wco", Page 2057
- Editing compensation tables during program run

  Further information: "Compensation during program run", Page 1974

Description of function

In order to compensate tools by using the compensation tables, the steps below are needed:

- Creating a compensation table

  Further information: "Creating a compensation table", Page 2058
- Activating the compensation table in the NC program

  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 1122
- As an alternative, activating the compensation table manually for the program run

  Further information: "Activating the compensation tables manually", Page 1122
- Activating a compensation value

  Further information: "Activating a compensation value with FUNCTION CORRDATA", Page 1123

The compensation table values can be edited within the NC program.

Further information: "Accessing table values ", Page 1991

The values in the compensation tables can be edited even while the program is running.

Further information: "Compensation during program run", Page 1974

Tool compensation in the tool coordinate system T-CS:

The compensation table ***.tco** defines compensation values for the tool in tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 1022

The compensations have the following effects:

- In the case of milling cutters, as an alternative to the delta values in the **TOOL CALL**

  Further information: "Tool call by TOOL CALL", Page 304
- In the case of turning tools, as an alternative to **FUNCTION TURNDATA CORR-TCS** (option 50)

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1124
- In the case of grinding tools, as compensation for **LO** and **R-OVR** (option 156)

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

The control displays an active shift using the compensation table ***.tco** in the **Tool** tab of the **Status** workspace.

Further information: "Tool tab", Page 184

Tool 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**).

Further information: "Working plane coordinate system WPL-CS", Page 1018

The ***.wco** compensation tables are used mainly for turning (option 50).

The compensations have the following effects:

- For turning operations, as an alternative to **FUNCTION TURNDATA CORR-WPL** (option 50)
- An X shift affects the radius

The following options are available for a shift in the WPL-CS:

- **FUNCTION TURNDATA CORR-WPL**
- **FUNCTION CORRDATA WPL**
- Shifting with the turning-tool table
  - Optional **WPL-DX-DIAM** column
  - Optional **WPL-DZ** column

> The shifts programmed with **FUNCTION TURNDATA CORR-WPL** and **FUNCTION CORRDATA WPL** are alternative programming options for the same shift.
>
> A shift in the working plane coordinate system (**WPL-CS**) defined by the turning-tool table is added to the **FUNCTION TURNDATA CORR-WPL** and **FUNCTION CORRDATA WPL** functions.

If a shift with the ***.wco** compensation table is active, the control displays it, including the path of the table on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 181

Activating the compensation tables manually

The compensation tables can be activated manually for the **Program Run** operating mode.

In the **Program Run** operating mode, the **Program settings** window contains the **Tables** area. In this area, a datum table and both compensation tables can be selected in one selection window for running the program.

When activating a table, the control will highlight this table with the status **M**.

17.4.1 Selecting a compensation table with SEL CORR-TABLE

Application

If you are using compensation tables, then use the function **SEL CORR-TABLE** to activate the desired compensation table from within the NC program.

Related topics

- Activating the compensation values in the table

  Further information: "Activating a compensation value with FUNCTION CORRDATA", Page 1123
- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 2055

  Further information: "Compensation table *.wco", Page 2057

Description of function

For the NC program, both a table ***.tco** and a table ***.wco** can be selected.

Input

11 SEL CORR-TABLE TCS "TNC:\table \corr.tco"	; Select compensation table **corr.tco**

The NC function includes the following syntax elements:

Syntax element	Meaning
SEL CORR-TABLE	Syntax initiator for selecting a compensation table
TCS or **WPL**	Compensation in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**
" " or **QS**	Path of table Fixed or variable name Selection by means of a selection window

17.4.2 Activating a compensation value with FUNCTION CORRDATA

Application

The **FUNCTION CORRDATA** function allows activating a row of the compensation table for the active tool.

Related topics

- Selecting a compensation table
  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 1122
- Contents of the compensation tables
  Further information: "Compensation table *.tco", Page 2055
  Further information: "Compensation table *.wco", Page 2057

Description of function

The activated compensation values are active up to the next tool change or until the end of the NC program.

If you change a value, then this change does not become active until the compensation is called again.

Input

11 FUNCTION CORRDATA TCS #1	; Activate row 1 of compensation table ***.tco**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION CORRDATA	Syntax initiator for activating a compensation value
TCS, **WPL** or **RESET**	Compensation in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS** or reset compensation
#, **" "** or **QS**	Desired table row Fixed or variable number or name Selection by means of a selection window Only when **TCS** or **WPL** are selected
TCS or **WPL**	Reset the compensation in **T-CS** or in **WPL-CS** Only if **RESET** has been selected

17.5 Compensating turning tools with FUNCTION TURNDATA CORR (option 50)

Application

With **FUNCTION TURNDATA CORR** you can define additional compensation values for the active tool. In the **TURNDATA CORR FUNCTION** you can enter delta values for tool lengths in the X direction **DXL** and in the Z direction **DZL**. The compensation values have an additive effect on the compensation values from the turning tool table.

The compensation can be defined in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**.

Further information: "Reference systems", Page 1010

Related topics

- Delta values in the turning tool table

  Further information: "Turning tool table toolturn.trn (option 50)", Page 2006
- Tool compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 1120

Requirement

- Combined milling/turning (software option 50)
- Required tool data defined for the tool type

  Further information: "Tool data for the tool types", Page 284

Description of function

The coordinate system in which the compensation is active can be defined:

- **FUNCTION TURNDATA CORR-TCS**: Tool compensation is active in the tool coordinate system
- **FUNCTION TURNDATA CORR-WPL**: Tool compensation is active in the workpiece coordinate system

With **FUNCTION TURNDATA CORR-TCS** you can define a cutter radius oversize **DRS**. This enables you to program an equidistant contour oversize. **DCW** allows you to compensate the recessing width of a recessing tool.

Tool compensation **FUNCTION TURNDATA CORR-TCS** is always active in the tool coordinate system, even during inclined machining.

FUNCTION TURNDATA CORR is always in effect for the active tool. A renewed **TOOL CALL** deactivates compensation. When you exit the NC program (e.g. with PGM MGT), the control automatically resets the compensation values.

Input

11 FUNCTION TURNDATA CORR-TCS:Z/X DZL:0.1 DXL:0.05 DCW:0.1	; Tool compensation in Z direction, X direction and for the width of the recessing tool

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA CORR	Syntax initiator for tool compensation of a turning tool
CORR-TCS:Z/X or **CORR-WPL:Z/X**	Tool compensation in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**
DZL:	Delta value for the tool length in Z direction Optional syntax element
DXL:	Delta value for the tool length in X direction Optional syntax element
DCW:	Delta value for the recessing tool width Only if **CORR-TCS:Z/X** was selected Optional syntax element
DRS:	Delta value for the cutter radius Only if **CORR-TCS:Z/X** was selected Optional syntax element

Note

During interpolation turning, the functions **FUNCTION TURNDATA CORR** and **FUNCTION TURNDATA CORR-TCS** are not active.

If you wish to compensate for a turning tool in Cycle **292 CONTOUR.TURNG.INTRP.**, then you must perform this in the cycle or in the tool table.

Further information: "Cycle 292 CONTOUR.TURNG.INTRP. (option 96)", Page 694

17.6 3D tool compensation (option 9)

17.6.1 Fundamentals

The control allows 3D tool compensation in CAM-generated NC programs with surface-normal vectors.

Further information: "Straight line LN", Page 1127

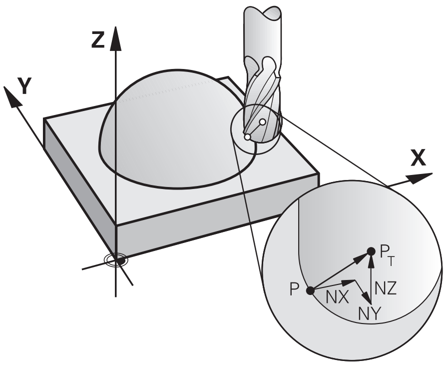
The control displaces the tool in the direction of the surface normals by the total of the delta values from tool management, tool call and compensation tables.

Further information: "Tools for 3D tool compensation", Page 1129

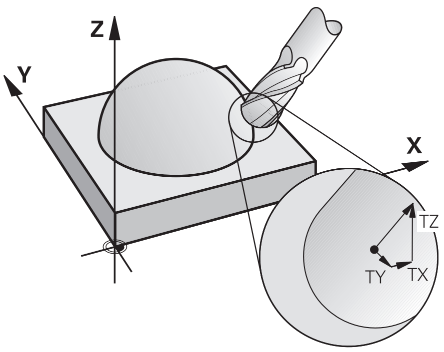
3D tool compensation can be used e. g. in the cases below:

- Compensation for re-worked tools for compensating small differences between the programmed and the actual tool dimensions
- Compensation for substitute tools with deviating diameters for compensating even larger differences between the programmed and the actual tool dimensions
- Generating a constant workpiece oversize which may serve as a finishing allowance, for example

3D tool compensation saves time since there is no need to recalculate and output from the CAM system.



> For an optional tool angle of inclination, the NC blocks must include an additional tool vector with the components TX, TY and TZ.



> Note the differences between face milling and peripheral milling.
>
> **Further information:** "3D tool compensation during face milling (option 9)", Page 1130
>
> **Further information:** "3D tool compensation during peripheral milling (option 9)", Page 1137

17.6.2 Straight line LN

Application

Straight lines **LN** are a prerequisite for 3D compensation. Within straight lines **LN**, a surface normal vector defines the direction of the 3D tool compensation. An optional tool vector defines the tool angle of inclination.

Related topics

- Fundamentals of 3D compensation

  Further information: "Fundamentals", Page 1126

Requirements

- Advanced Functions Set 2 (software option 9)
- NC program created with a CAM system

  Straight lines **LN** cannot be programmed directly on the control, but require a CAM system.

  Further information: "CAM-generated NC programs", Page 1301

Description of function

As with a straight line **L**, a straight line **LN** is used to define the target point coordinates.

Further information: "Straight line L", Page 326

In addition, the straight lines **LN** contain a surface normal vector as well as an optional tool vector.

Input

```
LN  X+31,737 Y+21,954 Z+33,165  NX+0,2637581 NY+0,0078922 NZ-0,8764339 TX
    +0,0078922 TY-0,8764339 TZ+0,2590319 F1000 M128
```

The NC function includes the following syntax elements:

Syntax element	Meaning
LN	Syntax initiator for straight line with vectors
X, **Y**, **Z**	Coordinates of the straight-line end point
NX, **NY**, **NZ**	Components of the surface normal vector
TX, **TY**, **TZ**	Components of the tool vector Optional syntax element
R0, **RL** or **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1114 Optional syntax element
F, **FMAX**, **FZ**, **FU** or **F AUTO**	Feed rate **Further information:** "Feed rate F", Page 310 Optional syntax element
M	Additional function Optional syntax element

Notes

- In the NC syntax, the order must be X,Y, Z for the position and NX, NY, NZ as well as TX, TY, TZ for the vectors.
- The NC syntax of LN blocks must always indicate all of the coordinates and all of the surface-normal vectors, even if the values have not changed from the previous NC block.
- Calculate the vectors as exactly as possible and specify them with at least 7 decimal places in order to avoid drastic feed rate decreases during machining.
- The CAM-generated NC program must contain normalized vectors.
- The 3D tool compensation using surface normal vectors is effective for the coordinate data specified for the main axes X, Y, Z.

Definition

Normalized vector
A normalized vector is a mathematical quantity possessing a magnitude of 1 and a direction. The direction is defined by the components X, Y and Z.

17.6.3 Tools for 3D tool compensation

Application

3D tool compensation can be used with the following tool shapes: end mill, toroid cutter and ball-nose cutter.

Related topics

- Compensation in tool management

  Further information: "Tool compensation for tool length and tool radius", Page 1112
- Compensation in tool call

  Further information: "Tool call by TOOL CALL", Page 304
- Compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 1120

Description of function

The tool shapes can be distinguished by columns **R** and **R2** of the tool management:

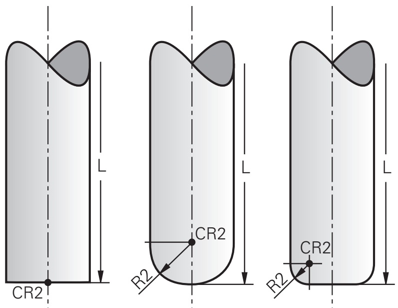
- End mill: **R2** = 0
- Toroid cutter: **R2** > 0
- Ball-nose cutter: **R2** = **R**

Further information: "Tool table tool.t", Page 1995

The delta values **DL**, **DR** and **DR2** are used to adapt the tool management values to the actual tool.

The control then compensates for the tool position by the sum of the delta values from the tool table and the programmed tool compensation (tool call or compensation table).

The surface normal vector of straight lines **LN** defines the direction in which the control compensates the tool. The surface normal vector always points to the tool radius 2 center CR2.



Position of CR2 with the individual tool shapes

Further information: "Presets on the tool", Page 271

Notes

- The tools are defined in the tool management. The overall tool length equals the distance between the tool carrier reference point and the tool tip. The control monitors the complete tool for collisions only by using the overall length.

  When defining a ball-nose cutter by the overall length and outputting an NC program to the ball center, the control must take the difference into account. When calling the tool in the NC program, define the sphere radius as a negative delta value in **DL** and thus shift the tool location point to the tool center point.
- If you load a tool with oversize (positive delta value), the control generates an error message. You can suppress the error message with the **M107** function.

  Further information: "Permitting positive tool oversizes with M107 (option 9)", Page 1357

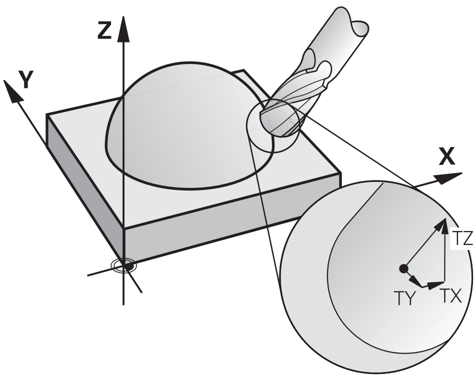
  Use the simulation to ensure that no contours are damaged by the tool oversize.

17.6.4 3D tool compensation during face milling (option 9)

Application

Face milling is a machining operation carried out with the front face of the tool.

The control displaces the tool in the direction of the surface normals by the total of the delta values from tool management, tool call and compensation tables.



Requirements

- Advanced Functions Set 2 (software option 9)
- Machine with automatically positionable rotary axes
- Output of surface normal vectors from the CAM system

  Further information: "Straight line LN", Page 1127
- NC program with **M128** or **FUNCTION TCPM**

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1341

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

Description of function

The variants below are possible with face milling:

- **LN** block without tool orientation, **M128** or **FUNCTION TCPM** is active: Tool perpendicular to the workpiece contour
- **LN** block with tool orientation **T**, **M128** or **FUNCTION TCPM** is active: Tool keeps the set tool orientation
- **LN** block without **M128** or **FUNCTION TCPM**: The control ignores the direction vector **T** even if it is defined

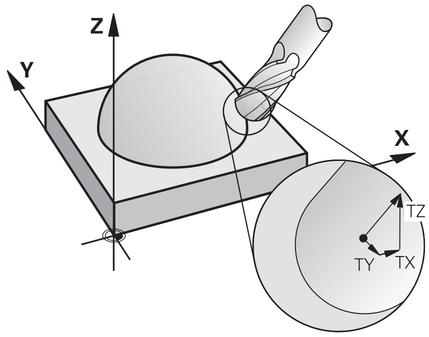
Example

11 L X+36.0084 Y+6.177 Z-1.9209 R0	; No compensation is possible
12 LN X+36.0084 Y+6.177 Z-1.9209 NX-0.4658107 NY+0 NZ+0.8848844 R0	; Compensation perpendicular to the contour is possible
13 LN X+36.0084 Y+6.177 Z-1.9209 NX-0.4658107 NY+0 NZ+0.8848844 TX +0.0000000 TY+0.6558846 TZ+0.7548612 R0 M128	; Compensation is possible, DL is effective along the T vector and DR2 along the N vector
14 LN X+36.0084 Y+6.177 Z-1.9209 NX-0.4658107 NY+0 NZ+0.8848844 R0 M128	; Compensation perpendicular to the contour is possible

Notes

NOTICE
Danger of collision! The rotary axes of a machine may have limited ranges of traverse (e.g., between –90° and +10° for the B head axis). Changing the tilt angle to a value of more than +10° may result in a 180° rotation of the table axis. There is a danger of collision during the tilting movement! ▶ Program a safe tool position before the tilting movement, if necessary. ▶ Carefully test the NC program or program section in the **Single Block** mode

- If no tool orientation was defined in the **LN** block, and **TCPM** is active, then the control maintains the tool perpendicular to the workpiece contour.

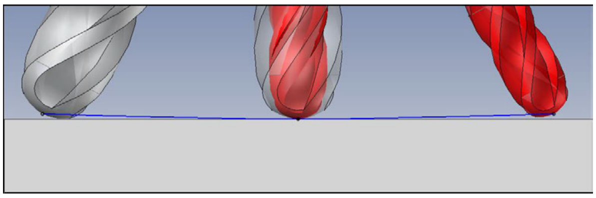


- If a tool orientation **T** is defined in the **LN** block, and **M128** (or **FUNCTION TCPM**) is simultaneously active, then the control automatically positions the rotary axes of the machine such that the tool reaches the defined tool orientation. If you have not activated **M128** (or **TCPM FUNCTION**), then the control ignores the direction vector **T,** even if it is defined in the **LN** block.
- The control is not able to automatically position the rotary axes on all machines.
- The control generally uses the defined **delta values** for 3D tool compensation. The entire tool radius (**R** + **DR**) is only taken into account if you have activated the **FUNCTION PROG PATH IS CONTOUR** function.

  Further information: "3D tool compensation with the entire tool radius with FUNCTION PROG PATH (option 9)", Page 1139

Examples

Compensate re-worked ball-nose cutter
CAM output at tool tip



Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

The NC program has the following structure:

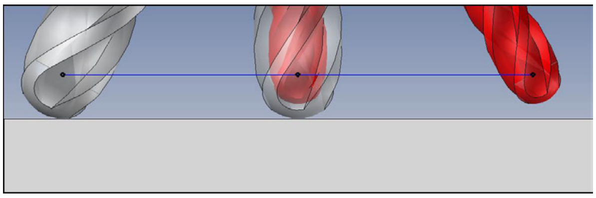
- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the tool tip
- Vector program with surface normal vectors

Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the tool table:
  - **R** and **R2** the theoretical tool data as from the CAM system
  - **DR** and **DR2** the difference between the nominal value and actual value

	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	-0.1	-0.1

Compensate re-worked ball-nose cutter
CAM output at the center of the ball



Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

The NC program has the following structure:

- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the center of the ball
- Vector program with surface normal vectors

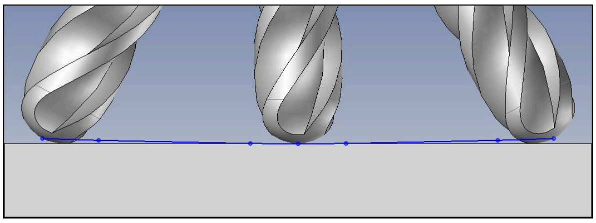
Suggested solution:

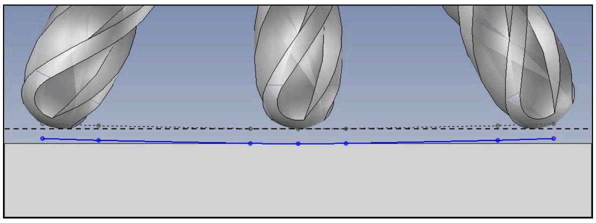
- Tool measurement on tool tip
- TCPM function **REFPNT CNT-CNT**
- Enter the tool compensation into the tool table:
  - **R** and **R2** the theoretical tool data as from the CAM system
  - **DR** and **DR2** the difference between the nominal value and actual value

	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	-0.1	-0.1

> With TCPM **REFPNT CNT-CNT** the tool compensation values are identical for the outputs on the tool tip or center of the ball.

Create workpiece oversize
CAM output at tool tip





Use a Ø 6 mm ball-nose cutter for achieving an even oversize of 0.2 mm on the contour.

The NC program has the following structure:

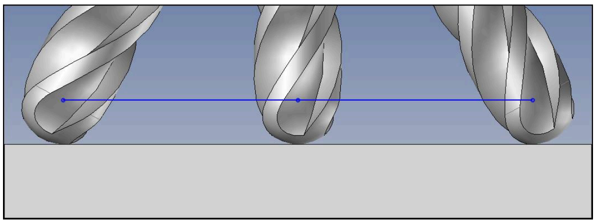
- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the tool tip
- Vector program with surface normal vectors and tool vectors

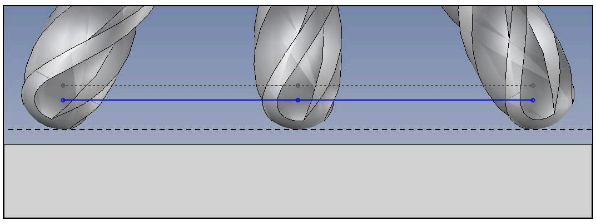
Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the TOOL CALL block:
  - **DL**, **DR** and **DR2** the desired oversize
- Suppress the error message with **M107**

	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	+0	+0
TOOL CALL			+0.2	+0.2	+0.2

Create workpiece oversize
CAM output at the center of the ball





Use a Ø 6 mm ball-nose cutter for achieving an even oversize of 0.2 mm on the contour.

The NC program has the following structure:

- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the center of the ball
- TCPM function **REFPNT CNT-CNT**
- Vector program with surface normal vectors and tool vectors

Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the TOOL CALL block:
  - **DL**, **DR** and **DR2** the desired oversize
- Suppress the error message with **M107**

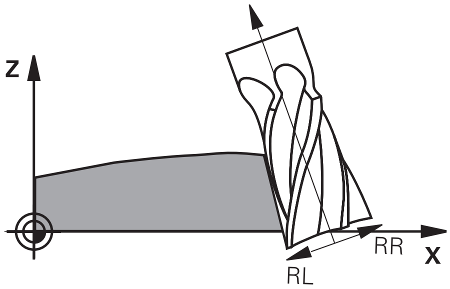
	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	+0	+0
TOOL CALL			+0.2	+0.2	+0.2

17.6.5 3D tool compensation during peripheral milling (option 9)

Application

Peripheral milling is a machining operation carried out with the lateral surface of the tool.

The control offsets the tool perpendicular to the direction of movement and perpendicular to the tool direction by the total of the delta values from the tool management, the tool call and the compensation tables.



Requirements

- Advanced Functions Set 2 (software option 9)
- Machine with automatically positionable rotary axes
- Output of surface normal vectors from the CAM system

  Further information: "Straight line LN", Page 1127
- NC program with spatial angles
- NC program with **M128** or **FUNCTION TCPM**

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1341

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
- NC program with tool radius compensation **RL** or **RR**

  Further information: "Tool radius compensation", Page 1114

Description of function

The variants below are possible with peripheral milling:

- **L** block with programmed rotary axes, **M128** or **FUNCTION TCPM** active, define compensation direction with radius compensation **RL** or **RR**
- **LN** block with tool orientation **T** perpendicular to the N vector, **M128** or **FUNCTION TCPM** is active
- **LN** block with tool orientation **T** without N vector, **M128**, or **FUNCTION TCPM** is active

Example

11 L X+48.4074 Y+102.4717 Z-7.1088 C-267.9784 B-20.0115 RL M128	; Compensation is possible, compensation direction RL
12 LN X+60.6593 Y+102.4690 Z-7.1012 NX0.0000 NY0.9397 NZ0.3420 TX-0.0807 TY-0.3409 TZ0.9366 R0 M128	; Compensation is possible
13 LN X+60.6593 Y+102.4690 Z-7.1012 TX-0.0807 TY-0.3409 TZ0.9366 M128	; Compensation is possible

Notes

NOTICE

Danger of collision!

The rotary axes of a machine may have limited ranges of traverse (e.g., between –90° and +10° for the B head axis). Changing the tilt angle to a value of more than +10° may result in a 180° rotation of the table axis. There is a danger of collision during the tilting movement!

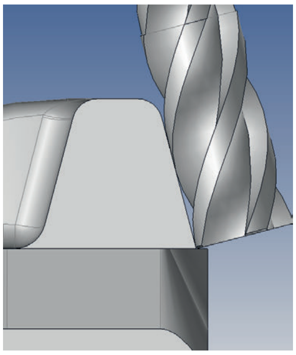
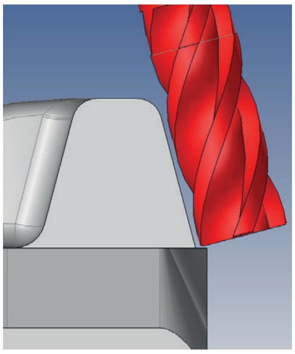
- Program a safe tool position before the tilting movement, if necessary.
- Carefully test the NC program or program section in the **Single Block** mode

- The control is not able to automatically position the rotary axes on all machines.
- The control generally uses the defined **delta values** for 3D tool compensation. The entire tool radius (**R** + **DR**) is only taken into account if you have activated the **FUNCTION PROG PATH IS CONTOUR** function.

  Further information: "3D tool compensation with the entire tool radius with FUNCTION PROG PATH (option 9)", Page 1139

Example

Compensate re-worked end mill
CAM output at tool center

You use a re-worked Ø 11.8 mm end mill instead of Ø 12 mm.

The NC program has the following structure:

- CAM output for Ø 12 mm end mill
- NC points output on the tool center
- Vector program with surface normal vectors and tool vectors

  Alternative:
- Klartext program with active tool radius compensation **RL**/**RR**

Proposed solution:

- Tool measurement on tool tip
- Suppress the error message with **M107**
- Enter the tool compensation into the tool table:
  - **R** and **R2** the theoretical tool data as from the CAM system
  - **DR** and **DL** the difference between the nominal value and the actual value

	R	R2	DL	DR	DR2
CAM	+6	+0			
Tool table	+6	+0	+0	-0.1	+0

17.6.6 3D tool compensation with the entire tool radius with FUNCTION PROG PATH (option 9)

Application

The **FUNCTION PROG PATH** function defines whether the control references the 3D radius compensation only to the delta values as in the past or to the entire tool radius.

Related topics

- Fundamentals of 3D compensation
  Further information: "Fundamentals", Page 1126
- Tools for 3D compensation
  Further information: "Tools for 3D tool compensation", Page 1129

Requirements

- Advanced Functions Set 2 (software option 9)
- NC program created with a CAM system
  Straight lines **LN** cannot be programmed directly on the control, but require a CAM system.
  Further information: "CAM-generated NC programs", Page 1301

Description of function

If you activate **FUNCTION PROG PATH**, the programmed coordinates exactly correspond to the contour coordinates.

The control takes the full tool radius **R + DR** and the full corner radius **R2 + DR2** into account for 3D radius compensation.

With **FUNCTION PROG PATH OFF**, you deactivate this special interpretation.

The control only uses the delta values **DR** and **DR2** for 3D radius compensation.

If you activate **FUNCTION PROG PATH**, the interpretation of the programmed path as the contour is active for 3D compensation movements until you deactivate the function.

Input

11 FUNCTION PROG PATH IS CONTOUR	; Use the entire tool radius for 3D compensation.

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION PROG PATH	Syntax initiator for interpreting the programmed path
IS CONTOUR or **OFF**	Use the entire tool radius or only the delta values for 3D compensation

17.7 3D radius compensation depending on the tool contact angle (option 92)

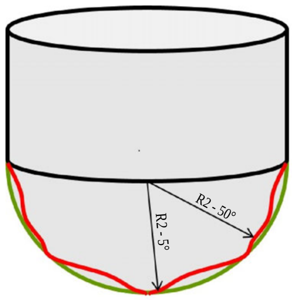
Application

Due to the production process, the effective spherical radius of a ball cutter deviates from the ideal form. The maximum form inaccuracy is defined by the tool manufacturer. Common deviations lie between 0.005 mm and 0.01 mm.

The form inaccuracy can be saved in the form of a compensation value table. This table contains angle values and the deviation from the nominal radius **R2** measured on the respective angle value.

The **3D-ToolComp** software (option 92) enables the control to compensate the value defined in the compensation value table depending on the actual contact point of the tool.

3D calibration of the touch probe can also be carried out with the **3D-ToolComp** software option. During this process the deviations determined during touch probe calibration are saved to the compensation value table.



Related topics

- Compensation value table *.3DTC

  Further information: "*.3DTC compensation table ", Page 2059
- Touch probe 3D calibration

  Further information: "Calibrating the workpiece touch probe", Page 1572
- 3D probing with a touch probe

  Further information: "Cycle 444 PROBING IN 3-D ", Page 1837
- 3D compensation with CAM-generated NC programs with surface-normal vectors

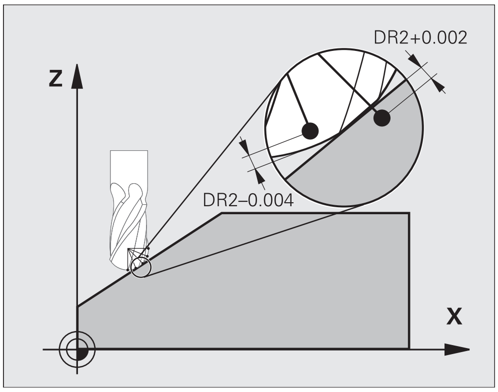
  Further information: "3D tool compensation (option 9)", Page 1126

Requirements

- Advanced Functions Set 2 (software option 9)
- 3D-ToolComp (software option 92)
- Output of surface normal vectors from the CAM system
- Tool properly defined in the tool management:
  - Value 0 in column **DR2**
  - Name of associated compensation value table in column **DR2TABLE**

  Further information: "Tool table tool.t", Page 1995

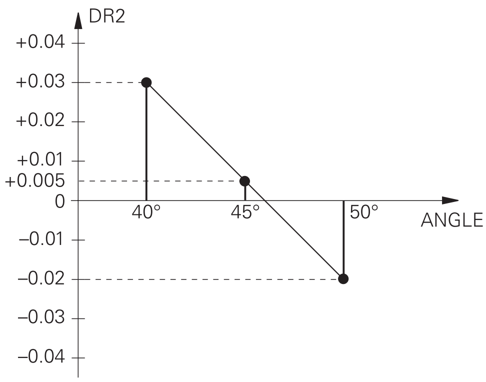
Description of function



If you are executing an NC program with surface-normal vectors and have assigned a compensation value table (DR2TABLE column) to the active tool in the tool table (TOOL.T), the control uses the values from the compensation value table instead of the compensation value DR2 from TOOL.T.

In doing so, the control takes the compensation value from the compensation value table defined for the current contact point of the tool with workpiece into account. If the contact point is between two compensation points, the control interpolates the compensation value linearly between the two closest angles.

Angle value	Compensation value
40°	0.03 mm (measured)
50°	-0.02 mm (measured)
45° (contact point)	+0.005 mm (interpolated)



Notes

- If the control cannot interpolate a compensation value, it displays an error message.
- **M107** (suppress error message for positive compensation values) is not required, even if positive compensation values are determined.
- The control uses either DR2 from TOOL.T or a compensation value from the compensation value table. Additional offsets, such as a surface oversize, can be defined via DR2 in the NC program (compensation table **.tco** or **TOOL CALL** block).

18

Files

18.1 File management

18.1.1 Basic information

Application

In the file management, the control displays drives, folders, and files. You can, for example, create or delete folders or files and can also connect drives.

The file management function encompasses the **Files** operating mode and the **Open File** window.

Related topics

- Data backup
  Further information: "Backup and restore", Page 2148
- Connecting network drives
  Further information: "Network drives on the control", Page 2113

Description of function

Icons and buttons

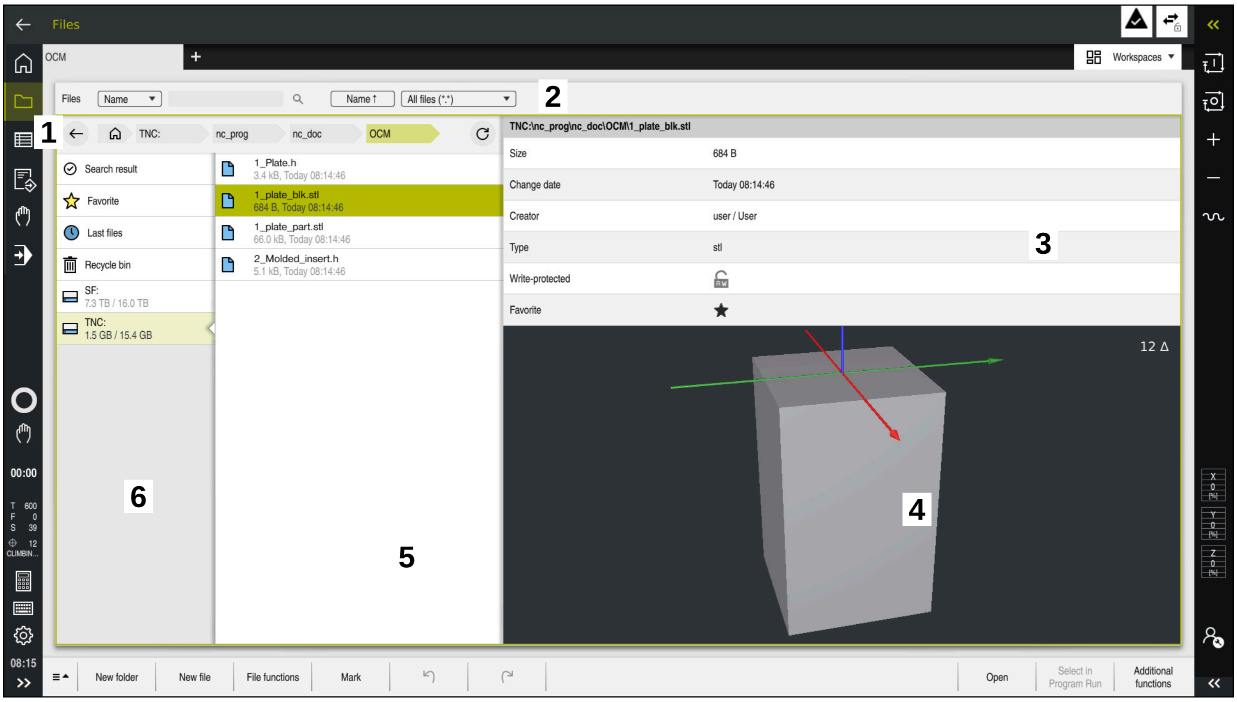
The file management contains the following icons and buttons:

Icon, button, or shortcut	Meaning
	Rename
CTRL+C	Copy
CTRL+X	Cut If you cut a file or a folder, then the control grays out the icon of the file or the folder.
	Delete
	Add a favorite
	Favorite If you add a favorite, then the control displays this symbol next to the file or the folder.
	Remove a favorite
	Eject USB device
	Activate write protection If write protection is active, then the control displays this symbol next to the file or the folder.
	Deactivate write protection
New folder	Create new folder

Icon, button, or shortcut	Meaning
New file	Create new file You create a new table in the **Tables** operating mode. **Further information:** "Tables operating mode", Page 1980
File functions	The control opens the context menu. **Further information:** "Context menu", Page 1522 Only in the **Files** operating mode
Mark **CTRL+BLANK**	The control marks the file and opens the action bar. Only in the **Files** operating mode
CTRL+Z	Undo an action
CTRL+Y	Redo an action
Open	The control opens the file in the appropriate operating mode or application.
Select in Program Run	The control opens the file in the **Program Run** operating mode. Only in the **Files** operating mode

Additional functions	The control opens a selection menu with the following functions:
	■ **Update TAB / PGM**
	  ■ Convert the format and content of files from the iTNC 530
	  ■ Modify faulty files
	  Further information: "Converting files", Page 1155
	■ **Mount network share**
	  Further information: "Network drives on the control", Page 2113
	Only in the **Files** operating mode

Screen elements of the file management



Files operating mode

1 Navigation path

In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level.

2 Title bar

- Full-text search

  Further information: "Full-text search in the title bar", Page 1147
- Sorting

  Further information: "Sorting in the title bar", Page 1147
- Filtering

  Further information: "Filtering in the title bar", Page 1147

3 Information area

Further information: "Information area", Page 1147

4 Preview area

In the preview area the control shows a preview of the selected file; for example an excerpt from an NC program.

5 Content column

In the content column the control shows all folders and files of the current drive, folder, or other source.

The control displays the following status for a file, if applicable:

- **M**: the file is active in the **Program Run** operating mode
- **S**: the file is active in the **Simulation** workspace
- **E**: the file is active in the **Editor** operating mode

6 Navigation column

Further information: "Navigation column", Page 1148

Full-text search in the title bar

Use the full-text search to look for any strings in the names or contents of files. The control searches through the active level and any lower levels of the selected drive or folder.

Use the selection menu to choose whether the control searches the names or contents of the files.

You can use the ***** character as a placeholder. This placeholder can stand for any characters or even an entire word. You can also use the placeholder to search for specific file types (e.g., ***.pdf**).

Sorting in the title bar

You can sort folders and files in ascending or descending order according to the following criteria:

- **Name**
- **Type**
- **Size**
- **Change date**

If you sort by name or type, the control lists the files alphabetically.

Filtering in the title bar

The control provides standard filters for file types. If you would like to filter for other file types, then you can search using the placeholder in the full-text search function.

Further information: "Full-text search in the title bar", Page 1147

Information area

In the information area the control shows the path of the file or folder.

Further information: "Path", Page 1148

Depending on which element is selected, the control displays the following additional information:

- **Size**
- **Change date**
- **Creator**
- **Type**

You can select the following functions in the information area:

- Activate and deactivate write-protection
- Add or remove favorites

Navigation column

The navigation column offers the following possibilities for navigation:

- **Search result**

  The control displays the results of the full-text search. If there was no search, or if nothing was found, then this area is empty.
- **Favorite**

  The control displays all folders and files that you have marked as favorites.
- **Last files**

  The control displays the 15 most recently opened files.
- **Recycle bin**

  The control moves deleted folders and files to the recycle bin. You can use the context menu to restore these files or empty the recycle bin.

  Further information: "Context menu", Page 1522
- Drives (e.g., **TNC:**)

  The control displays internal and external drives (e.g., a USB device).

  The control displays the occupied and total memory space under each drive.

Permitted characters

You can use the following characters for the names of drives, folders, and files:

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 _ -

Only use characters that are shown here; otherwise problems might occur (for example, during data transmission).

The following characters have specific functions, and must therefore not be used in a name:

Symbol	Function
.	Separates the file name from the file type
\ /	Separates between drive, folder, and file in the path
:	Separates the drive names

Name

When you create a file, you first define its name. The file name is followed by the file name extension, consisting of a period and the file type.

Path

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

Absolute path

An absolute path specifies the exact position of a file. The path begins with the drive and then goes through the folder structure in sequence all the way to the file (e.g., **TNC:\nc_prog\$mdi.h**). If the file being called has been moved, then a new absolute path must be entered.

Relative path

A relative path specifies the position of a file in relation to the file that is calling it. The path goes through the folder structure in sequence all the way to the file, starting from the file that is calling it (e.g., **demo\reset.H**). If a file has been moved, then a new relative path must be entered.

File types

You can use uppercase or lowercase letters to define the file type.

HEIDENHAIN-specific file types

The control can open the following HEIDENHAIN-specific file types:

File type	Application
H	NC program written in HEIDENHAIN Klartext **Further information:** "Contents of an NC program", Page 212
I	NC program with ISO commands
HC	Contour definition in the smarT.NC format of the iTNC 530
HU	Main program in the smarT.NC format of the iTNC 530
3DTC	Table with 3D tool compensations that depend on the contact angle **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1140
D	Table with workpiece datums **Further information:** "Datum table", Page 2045
DEP	Automatically generated table with data that depend on the NC program (e.g., the tool usage file) **Further information:** "Tool usage file", Page 2029
P	Table for pallet-oriented machining **Further information:** "Job list workspace", Page 1938
PNT	Table with machining positions (e.g., for the machining of irregular point patterns) **Further information:** "Point table", Page 2044
PR	Table with workpiece presets **Further information:** "Preset table", Page 2035
TAB	Freely definable table (e.g., for protocol files or as WMAT and TMAT tables for automatic calculation of cutting data) **Further information:** "Freely definable tables", Page 2034 **Further information:** "Cutting data calculator", Page 1529
TCH	Table with the assignment of the tool magazine **Further information:** "Pocket table tool_p.tch", Page 2026
T	Table with tools for all technologies **Further information:** "Tool table tool.t", Page 1995
TP	Table with touch probes **Further information:** "Touch probe table tchprobe.tp", Page 2022
TRN	Table with turning tools **Further information:** "Turning tool table toolturn.trn (option 50)", Page 2006
GRD	Table with grinding tools **Further information:** "Grinding tool table toolgrind.grd (option 156)", Page 2010

File type	Application
DRS	Table with dressing tools **Further information:** "Dressing tool table tooldress.drs (option 156)", Page 2019
TNCDRW	Contour description as a 2D drawing **Further information:** "Graphical Programming", Page 1437
M3D	Format for tool carriers or collision objects (option 40), for example **Further information:** "Options for fixture files", Page 1172
TNCBCK	File for data backup and restoration **Further information:** "Backup and restore", Page 2148
EXP	Configuration file for saving and importing configurations of the control interface **Further information:** "Configuring the control's user interface", Page 2157

The control opens these file types with an internal application or with a HEROS tool.

Further information: "Opening files with additional software", Page 2194

Standardized file types

The control can open the following standardized file types:

File type	Application
CSV	Text file for saving or exchanging simple structured data **Further information:** "Importing and exporting tool data", Page 298
XLSX (XLS)	File type for various spreadsheet programs (e.g., Microsoft Excel)
STL	3D model created with triangular facets (e.g., fixtures) **Further information:** "Exporting a simulated workpiece as STL file", Page 1546
DXF	2D CAD files
IGS/IGES STP/STEP	3D CAD files **Further information:** "Opening CAD Files with the CAD-Viewer", Page 1455
CHM	Help files in compiled or compressed format
CFG	Configuration files of the control **Further information:** "Options for fixture files", Page 1172 **Further information:** "Machine parameters", Page 2152
CFT	3D data of a parameterizable tool-carrier template **Further information:** "Tool carrier management", Page 301
CFX	3D data of a geometrically determined tool carrier **Further information:** "Tool carrier management", Page 301
HTM/HTML	Text file with structured content of a website that can be opened in a browser (e.g., the integrated product help) **Further information:** "User's Manual as integrated product aid: TNCguide", Page 82
XML	Text file with hierarchically-structured data
PDF	Document format that visually reproduces the original file identically, regardless of the source application
BAK	Data-backup file **Further information:** "Data backup", Page 2194
INI	Initialization file (e.g., can contain program settings)
A	Text file (e.g., for defining the screen output format with FN16)
TXT	Text file (e.g., for saving the results of measurement cycles with FN16)
SVG	Picture format for vector graphics
BMP GIF JPG/JPEG PNG	Picture formats for pixel graphics By default, the control uses the PNG format for screenshots **Further information:** "HEROS menu", Page 2184
OGG	Container file format for the OGA, OGV, and OGX media types
ZIP	Container file format that collects multiple compressed files.

The control opens some of these file types with the HEROS tools.

Further information: "Opening files with additional software", Page 2194

Notes

- The control has 189 GB of disk space. The maximum size of any file is limited to 2 GB.
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). These characters can cause problems when inputting or reading data in conjunction with SQL commands.

  Further information: "Table access with SQL statements", Page 1417
- If the cursor is within the content column, you can start inputting through the keyboard. The control opens a separate input field and automatically searches for the entered string. If it finds a file or folder with that string, then the control moves the cursor to it.
- If you exit an NC program with the **END BLK** key, then the control opens the **Add** tab. The cursor is on the NC program that was just closed.

  If you press the **END BLK** key again, the control opens the 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 control creates dependency files with the ***.dep** extension for the tool-usage file (e.g., in order to perform a tool usage test).

  Further information: "Tool usage test", Page 312

  In the machine parameter **dependentFiles** (no. 122101) the machine manufacturer defines whether the control displays dependency files.
- In the machine parameter **createBackup** (no. 105401) the machine manufacturer defines whether the control creates a backup file when saving an NC program. Please note that these backup files will take up disk space.

Hints about file functions

If you select a file or folder and swipe to the right, the control displays the following file functions:

- Rename
- Copy
- Cut
- Delete
- Activate or deactivate write protection
- Add or remove a favorite

You can also select some of these file functions with the context menu.

Further information: "Context menu", Page 1522

Hints about copied files

- If you copy a file and then paste it to the same folder, the control adds the suffix **_Copy** to the file name.
- If you paste a file to another folder and that folder contains a file with the same name, the control opens the **Insert file** window. The control displays the path of the two files and offers the following options:
  - Replace existing file
  - Skip copied file
  - Add suffix to file name

  You can also apply the selected option to all such cases.

18.1.2 Open File workspace

Application

In the **Open File** workspace you can select and create files, for example.

Description of function

The **Open File** workspace can be opened by the icons below, depending on the active operating mode:

Icon	Function
	**Add** in the **Tables** and **Editor** operating modes
	**Open File** in the **Program Run** operating mode

The functions below can be executed in the **Open File** workspace in the respective operating modes:

Function	Tables operating mode	Editor operating mode	Program Run operating mode
New folder	✓	✓	–
New file	✓	✓	–
Open	✓	✓	✓

18.1.3 Quick selection workspace

Application

In the **Quick selection** workspace, you can create files or open existing ones regardless of the active operating mode.

Description of function

You can open the **Quick selection** workspace with the **Add** function in the following operating modes:

- **Tables**

  Further information: "Quick selection workspace in the Tables operating mode", Page 1154
- **Editor**

  Further information: "Quick selection workspace in the Editor operating mode", Page 1154

Further information: "Icons on the control's user interface", Page 123

Quick selection workspace in the Tables operating mode

In the **Tables** operating mode, the **Quick selection** workspace offers the following buttons:

- **Create new table**
- **Tool management**
- **Pocket table**
- **Presets**
- **Touch probes**
- **Datums**
- **T usage order**
- **Tooling list**

The **Quick selection** workspace contains the following areas:

- **Active tables for machining**
- **Active tables for simulation**

The control displays the **Presets** and **Datums** buttons in both areas.

With the **Presets** and **Datums** buttons, you can open the table that is active in the program run or in the simulation. If the same table is active in program run and the simulation, then the control opens this table only once.

Quick selection workspace in the Editor operating mode

The **Quick selection** workspace offers the following buttons in the **Editor** operating mode:

- **New program mm**
- **New program inch**
- **New DIN/ ISO program mm**
- **New DIN/ ISO program inch**
- **New contour**
- **New job list**

18.1.4 Document workspace

Application

In the **Document** workspace, you can open files in order to view them, such as a technical drawing.

Related topics

- Supported file types
  Further information: "File types", Page 1149

Description of function

The **Document** workspace is available in every operating mode and application. If you open a file, then the control displays the same file in all operating modes.

Further information: "Overview of operating modes", Page 110

You can open the following file types in the **Document** workspace:

- PDF files
- HTML files
- Text files, such as *.a
- Image files, such as *.png
- Video files, such as *.ogg

Further information: "File types", Page 1149

You can, for example, transfer dimensions from a technical drawing using the clipboard in the NC program.

Open file

To open the file in the **Document** workspace:

- If applicable, open the **Document** workspace

- Select **Open File**
- The control opens a selection window with the file manager.
- Select the desired file

Open

- Select **Open**
- The control displays the file in the **Document** workspace.

18.1.5 Converting files

Application

In order to use a file created on the iTNC 530 on the **TNC7** as well, the control must adapt the file's format and content. The **Update TAB / PGM** function does this job.

Description of function

Importing an NC program

The control uses the **Update TAB / PGM** function to remove umlauts and checks if the NC block **END PGM** exists. The NC program would be incomplete without this NC block.

Importing a table

The following characters are permitted in the **NAME** column of the tool table:
$ % & , - . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z _

If you convert tables from an earlier control using the **Update TAB / PGM** function, then the control makes the following changes as needed:

- The control changes decimal commas into decimal points.
- The control adopts all supported tool types and assigns the **Undefined** type to all unknown tool types.

The **Update TAB / PGM** function also allows you to convert tables of the TNC7 if necessary.

Further information: "Tool table tool.t", Page 1995

Adapting a file

Prepare a backup of the original file before adapting

To adapt the format and the content of an iTNC 530 file:

- ▶ Select the **Files** operating mode
- ▶ Select the desired file
- ▶ Select **Additional functions**

Additional functions

- > The control displays a selection menu.
- ▶ Select **Update TAB / PGM**
- > The control adapts the file format and content.

> The control saves the changes and overwrites the original file.

- ▶ Check the content after adapting

Notes

NOTICE

Caution: Data may be lost!

If you use the **Update TAB / PGM** function, then data may be irrevocably deleted or altered!

- ▶ Create a backup copy prior to converting the file

- The machine manufacturer uses import and update rules to define which adaptations the control is to execute, e. g. removal of umlauts.
- The machine manufacturer uses the optional machine parameter **import-FromExternal** (no. 102909) to define for each file type if automatic adaptation is carried out upon copying to the control.

18.1.6 USB devices

Application

A USB device allows transmitting data and saving data externally.

Requirement

- USB 2.0 or 3.0
- USB device with supported file system

  The control supports USB devices with the following file systems:
  - FAT
  - VFAT
  - exFAT
  - ISO9660

  > The control does not support USB devices with other file systems, such as NTFS.

- A ready data interface

  Further information: "Serial data transfer", Page 2189

Description of function

The control displays a USB device as a drive in the navigation column of the **Files** operating mode or of the **Open File** workspace.

The control automatically detects USB devices. If you connect a USB device with a file system that is not supported, the control generates an error message.

Before executing an NC program saved on the USB device, the file must be transferred to the control hard disk.

When transmitting large files, the control displays the data transmission progress at the bottom of the navigation and content column.

Removing a USB device

To remove a USB device:

- ▶ Select **Eject**
- \> The control opens a pop-up window and asks whether you want to eject the USB device.

OK

- ▶ Press **OK**
- \> The control shows the message **The USB device can be removed now**.

Notes

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- Copy the NC program and all called files to the **TNC:** drive

NOTICE

Caution: Data may be lost!

Always remove a connected USB device properly, otherwise data may be damaged or deleted!

- Use the USB port for transfer and backup only; do not use it for editing and executing NC programs
- Use the icon to remove USB devices when data transfer is complete

- If an error message is displayed when connecting a USB device, check the setting in the **SELinux** security software.

  Further information: "SELinux security software", Page 2112
- If the control displays an error message when using a USB hub, ignore and acknowledge the message with the **CE** key.
- Prepare a backup of the files on the control at regular intervals.

  Further information: "Data backup", Page 2194

18.2 Programmable file functions

Application

Programmable file functions enable management of files from within the NC program. Files can be opened, copied, relocated and deleted. This permits, for example, opening the drawing of a component during the measuring process with a touch probe cycle.

Description of function

Opening a file with OPEN FILE

The **OPEN FILE** function allows you to open a file from within an 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: "File types", Page 1149

The control opens the file in the HEROS tool last used for this file type. If you have never opened a file of a certain file type and multiple HEROS tools are available, the control will interrupt program run and open the **Application?** window. In the **Application?** window, you can select the HEROS tool the control should use to open the file. The control saves this selection.

Multiple HEROS 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 HEROS 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 HEROS tools, you can use the file manager to select the HEROS tool to be used for opening files of this file type.

Further information: "File management", Page 1144

Input

```
11 OPEN FILE "FILE1.PDF" STOP
```

The NC function includes the following syntax elements:

Syntax element	Meaning
OPEN FILE	Start of syntax for the OPEN FILE function
" "	Path of the file to be opened
STOP	Interrupts the program run or simulation Optional syntax element

Copying, relocating and deleting files with FUNCTION FILE

The control offers the functions below for copying, moving and deleting files from an NC program:

NC function	Description
FUNCTION FILE COPY	This function copies a file into a target file. The control substitutes the content of the target file. This function requires specifying the path to both files.
FUNCTION FILE MOVE	This function moves a file to a target file. The control substitutes the content of the target file and deletes the file to be moved. This function requires specifying the path to both files.
FUNCTION FILE DELETE	This function deletes the selected file. This function requires specifying the path to the file to be deleted.

Input

11 FUNCTION FILE COPY "FILE1.PDF" TO "FILE2.PDF"	; Copy the file from the NC program

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE COPY	Syntax initiator for the Open file function
" "	Path of the file to be copied
" "	Path of the file to be substituted

11 FUNCTION FILE MOVE "FILE1.PDF" TO "FILE2.PDF"	; Move the file from the NC program

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE MOVE	Syntax initiator for the Move file function
" "	Path of the file to be relocated
" "	Path of the file to be substituted

11 FUNCTION FILE DELETE "FILE1.PDF"	; Delete the file from the NC program

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE DELETE	Syntax initiator for the Delete file function
" "	Path of the file to be deleted

Notes

NOTICE

Caution: Data may be lost!

When deleting a file with the **FUNCTION FILE DELETE** function, the control will not put this file into the recycle bin. The control deletes the file once and for all!

- Use this function only with files that are no longer needed

- There are various ways to select files:
  - Enter the file path
  - Select the file in a select window
  - Define the file path or name of the subprogram in a QS parameter

    If the called file is located in the same directory as the calling file, you may also enter just the file name.
- When applying file functions relating to the calling NC program in a called NC program, the control will display an error message.
- When intending to copy or move a non-existent file, the control displays an error message.
- If the file to be deleted does not exist, the control does not display an error message.

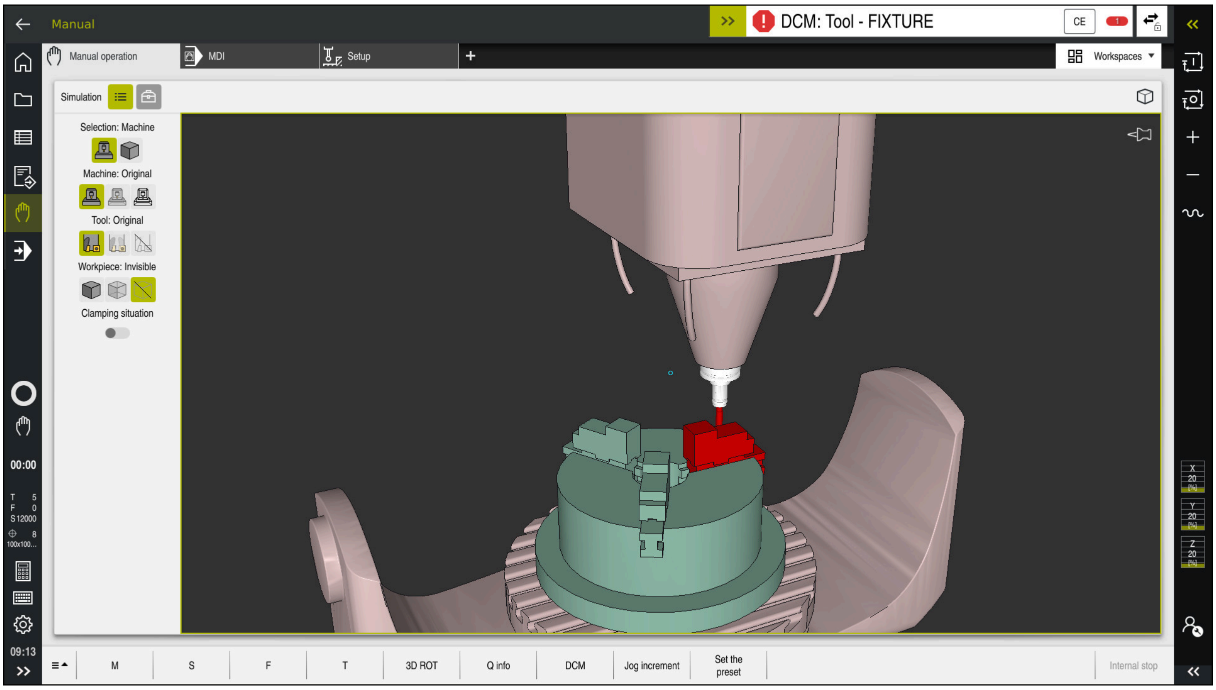
19

Collision Monitoring

19.1 Dynamic Collision Monitoring (DCM, option 40)

Application

Dynamic Collision Monitoring (DCM, dynamic collision monitoring) can be used for collision monitoring of machine components defined by the machine manufacturer. When the collision objects come closer to each other than a defined minimum distance, the control stops and displays an error message. This procedure reduces the risk of collision.



Dynamic Collision Monitoring (DCM) including collision warning

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- Control prepared by the machine manufacturer

  The machine manufacturer must define a kinematics model of the machine, insertion point for fixtures and the safety distance between collision objects.

  Further information: "Fixture monitoring (option 40)", Page 1171
- Tools with a positive radius **R** and length **L**.

  Further information: "Tool table tool.t", Page 1995
- The values in the tool management equal the actual tool dimensions

  Further information: "Tool management ", Page 297

Description of function

> Refer to your machine manual.
> The machine manufacturer adapts Dynamic Collision Monitoring (DCM) to the control.

The machine manufacturer can define machine components and minimum distances that are to be monitored by the control during all machine movements. If two collision objects come closer to each other than a defined minimum distance, the control generates an error message and terminates the movement.

Error message for Dynamic Collision Monitoring (DCM)

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

The control displays the collision objects graphically in the following operating modes:

- **Editor** operating mode
- **Manual** operating mode
- **Program Run** operating mode

The control also monitors the tools, as defined in tool management, for collision.

NOTICE

Danger of collision!

Even if Dynamic Collision Monitoring (DCM) is active, the control does not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

- Enable the **Advanced checks** toggle switch for simulations
- Check the machining sequence using a simulation
- Carefully test your NC program or program section in the **Single Block** mode

Further information: "Advanced checks in the simulation", Page 1190

Dynamic Collision Monitoring (DCM) in the Manual and Program Run operating modes

Dynamic Collision Monitoring (DCM) is activated separately for the **Manual** and **Program Run** operating modes, using the **DCM** button.

Further information: "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 1168

In the **Manual** and **Program Run** operating modes, the control stops the movement if two collision objects approach each other by less than a minimum distance. In this case, the control displays an error message naming the two objects causing collision.

⚙	Refer to your machine manual. The machine manufacturer can define the minimum distance between two collision-monitored objects.

Before the collision warning, the control dynamically reduces the feed rate of movements. This ensures that the axes stop in good time before a collision occurs.

When the collision warning is triggered, the control displays the colliding objects in red in the **Simulation** workspace.

ℹ	When a collision warning has been issued, machine movements via the axis direction keys or the handwheel are only possible if they increase the distance between the collision objects. With active collision monitoring and a simultaneous collision warning, no movements are permitted that reduce the distance or leave it unchanged.

Dynamic Collision Monitoring (DCM) in the Editor operating mode

Dynamic Collision Monitoring (DCM) is activated for simulation in the **Simulation** workspace.

Further information: "Activating Dynamic Collision Monitoring (DCM) for the simulation", Page 1168

In the **Editor** operating mode, an NC program can be collision-monitored even prior to execution. In case of collision, the control stops the simulation and displays an error message naming the two objects causing collision.

HEIDENHAIN recommends the use of Dynamic Collision Monitoring (DCM) in the **Editor** operating mode only in addition to DCM in the **Manual** and **Program Run** operating modes.

> The enhanced collision monitoring shows collisions between the workpiece and tools or tool holders.
>
> **Further information:** "Advanced checks in the simulation", Page 1190

To obtain a simulation result that is similar to the program run, the following aspects must match:

- Workpiece preset
- Basic rotation
- Offsets of each axis
- Tilting condition
- Active kinematic model

The active workpiece preset for the simulation must be selected. The active workpiece preset from the preset table can be adopted into the simulation.

Further information: "Visualization options column", Page 1537

In a simulation, the following aspects may differ from the actual machine or may not be available at all:

- The simulated tool change position may differ from the tool change position in the machine.
- Changes in the kinematics may have a delayed effect in the simulation.
- PLC positioning movements are not displayed in the simulation.
- Global program settings (GPS, option 44) are not available
- Handwheel superimposition is not available
- Editing of job lists is not available
- Traverse range limits from the **Settings** application are not available.

19.1.1 Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

To active Dynamic Collision Monitoring (DCM) for the **Manual** and **Program Run** operating modes:

- Select the **Manual** operating mode
- Select the **Manual** application

DCM

- Select **DCM**
- > The control opens the **Dyna. Coll. Monitoring (DCM)** window.
- Activate DCM in the desired operating modes, using the toggle switches

OK

- Press **OK**
- > The control activates DCM in the selected operating modes.

The control displays the status of Dynamic Collision Monitoring (DCM) in the **Positions** workspace. When deactivating DCM, the control displays an icon in the information bar.

19.1.2 Activating Dynamic Collision Monitoring (DCM) for the simulation

Dynamic Collision Monitoring (DCM) can be activated for the simulation only in the **Editor** operating mode.

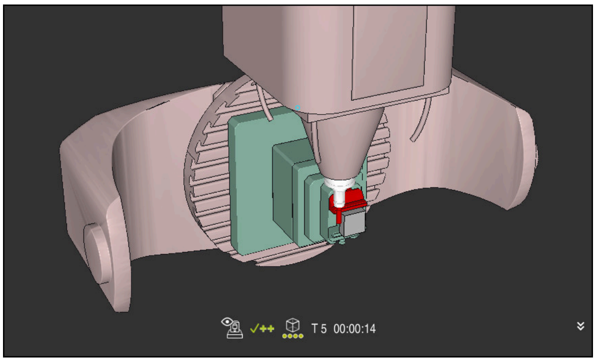
To activate DCM for the simulation:

- Select the **Editor** operating mode
- Select **Workspaces**
- Select **Simulation**
- > The control opens the **Simulation** workspace.
- Select the **Visualization options** column
- Activate the **DCM** toggle switch
- > The control activates DCM in the **Editor** operating mode.

The control displays the status of Dynamic Collision Monitoring (DCM) in the **Simulation** workspace

Further information: "Icons in the Simulation workspace", Page 1536

19.1.3 Activating the graphic display of the collision objects



Simulation in the **Machine** mode

To activate the graphic display of the collision objects:

- Select an operating mode (e.g., **Manual**)
- Select **Workspaces**
- Select the **Simulation** workspace
- > The control opens the **Simulation** workspace.
- Select the **Visualization options** column
- Select the **Machine** mode
- > The control displays a graphic representation of the machine and the workpiece.

Changing the representation

To change the graphic display of the collision objects:

- Activate the graphic display of the collision objects
- Select the **Visualization options** column
- Change the graphic display of the collision objects (e.g., **Original**)

19.1.4 FUNCTION DCM: Deactivating and activating Dynamic Collision Monitoring (DCM) in NC programs

Application

Some machining steps are by design performed close to a collision object. If you want to exclude some machining steps from Dynamic Collision Monitoring (DCM), you can deactivate DCM for them in your NC program. This means that it is possible to monitor individual parts of an NC program for collision.

Requirement

This function can only be used if Dynamic Collision Monitoring (DCM) is active for the **Program Run** operating mode. Otherwise, the function has no effect and you cannot activate DCM from within the NC program.

Description of function

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

FUNCTION DCM is only effective within the NC program.

It is for example possible to deactivate Dynamic Collision Monitoring (DCM) in the following situations in your NC program:

- To reduce the distance between two objects monitored for collision
- To prevent stops during program runs

The following NC functions are available:

- **FUNCTION DCM OFF** deactivates collision monitoring until the end of the NC program or the call of the **FUNCTION DCM ON** function.
- **FUNCTION DCM ON** revokes the **FUNCTION DCM OFF** function and reactivates collision monitoring.

Programming the FUNCTION DCM function

To program the **FUNCTION DCM** function:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **FUNCTION DCM**
- Select the **OFF** or **ON** syntax element

Notes

- Dynamic Collision Monitoring (DCM) helps you reduce the risk of collision. However, the control cannot consider all possible constellations during operation.
- The control can protect only those machine components from collision that your machine manufacturer has defined correctly with regard to dimensions, orientation, and position.
- The control takes the **DL** and **DR** delta values from the tool management into account. Delta values from the **TOOL CALL** block or a compensation table are not taken into account.
- For certain tools (e.g., face-milling cutters) the radius that would cause a collision can be greater than the value defined in the tool management.
- When a touch probe cycle starts, the control no longer monitors the stylus length and ball-tip diameter, so you can still probe collision objects.

19.2 Fixture monitoring (option 40)

19.2.1 Fundamentals

Application

The Fixture Monitoring function allows you to map setup situations and monitor them for collisions.

Related topics

- Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
- Integrating an STL file as workpiece blank

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 264

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- Kinematics description

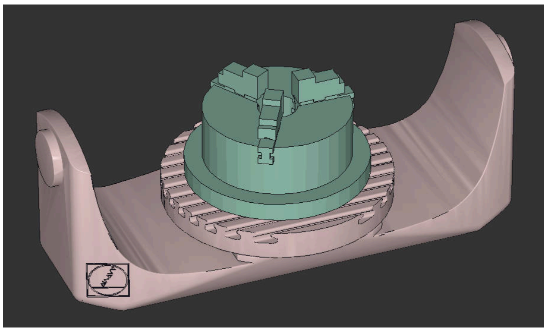
  The machine manufacturer creates the kinematics description
- Insertion point defined

  Using the insertion point, the machine manufacturer defines the preset for positioning the fixtures. The insertion point is often located at the end of the kinematic chain (e.g., at the center of a rotary table). For information about the position of the insertion point, please refer to your machine manual.
- Fixtures of suitable format:
  - STL file
    - 20,000 triangles maximum
    - Triangular mesh forms a closed shell
  - CFG file
  - M3D file

Description of function

To use fixture monitoring, the steps below are needed:

- Creating a fixture or loading into the control
  Further information: "Options for fixture files", Page 1172
- Fixture placement
  - **Set up fixtures** function in the **Setup** application (option 140)
    Further information: "Integrating the fixtures into collision monitoring (option 140)", Page 1174
  - Manual fixture placement
- When changing fixtures, load or remove the fixture in the NC program
  Further information: "Loading and removing fixtures using the FIXTURE function (option 40)", Page 1183



Three-jaw chuck loaded as fixture

Options for fixture files

For the integration of fixtures with the **Set up fixtures** function, only STL files can be used.

You can use the **3D mesh** function (option 152) to create STL files from other file types and adapt STL files to the requirements of your control.

Further information: "Generating STL files with 3D mesh (option 152)", Page 1472

Alternatively, CFG and M3D files can be set up manually.

Fixtures from STL files

STL files allow you to map both individual components and entire assemblies as an immobile fixture. The STL format is useful, in particular, for datum clamping systems and recurring setups.

If an STL file does not meet the requirements of the control, then the control issues an error message.

With CAD Model Optimizer (software option 152), you can adapt STL files that do not meet the requirements and then use them as fixtures.

Further information: "Generating STL files with 3D mesh (option 152)", Page 1472

Fixtures from M3D files

M3D is a file type designed by HEIDENHAIN. The paid M3D Converter software from HEIDENHAIN allows you to create M3D files from STL or STEP files.

In order to use an M3D file as a fixture, you need to use the M3D Converter software to create and check the file.

Fixtures from CFG files

CFG files are configuration files. You can integrate the STL and M3D files available in a CFG file. This enables you to map complex setups.

The **Set up fixtures** function can be used to create a CFG file for the fixture, using the measured value.

In CFG files, you can correct the orientation of the fixture files to be effective on the control. **KinematicsDesign** can be used to create and edit CFG files on the control.

Further information: "Editing CFG files with KinematicsDesign", Page 1184

Notes

NOTICE

Danger of collision!

The setup situation defined for fixture monitoring must match the actual machine status. Otherwise, there is a risk of collision.

- Measure the position of the fixture in your machine
- Use the measured values for positioning the fixture
- Test the NC programs in the Simulation

- When using a CAM system, use a postprocessor to output the fixture situation.
- Note the orientation of the coordinate system in the CAD system. Use the CAD system to adapt the orientation of the coordinate system to the desired orientation of the fixture in the machine.
- You can choose any orientation of the fixture model in the CAD system, and therefore the orientation does not always match the orientation of the fixture in the machine.
- Define the coordinate origin in the CAD system such that the fixture can be directly attached to the point of insertion of the kinematics.
- Create a central directory for your fixtures (e.g., **TNC:\system\Fixture**).
- HEIDENHAIN recommends storing variants of recurring setup situations suitable for standard workpiece sizes in the control (e.g., vise with different jaw opening widths).

  By storing multiple fixtures, you can choose the appropriate fixture for your machining operation without needing to configure it.
- Example files for setups used in everyday manufacturing are provided in the NC database of the Klartext Portal:

  https://www.klartext-portal.com/en/tips/nc-solutions

19.2.2 Integrating the fixtures into collision monitoring (option 140)

Application

The **Simulation** function determines the position of a 3D model in the **Set up fixtures** workspace, matching the real fixture in the machine envelope. Once the fixture has been set-up, the control considers it in Dynamic Collision Monitoring (DCM).

Related topics

- **Simulation** workspace
  Further information: "Simulation Workspace", Page 1535
- Dynamic Collision Monitoring (DCM)
  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
- Fixture monitoring
  Further information: "Fixture monitoring (option 40)", Page 1171
- Setting up the workpiece with graphic support (option 159)
  Further information: "Setting up the workpiece with graphical support (option 159)", Page 1582

Requirements

- Dynamic Collision Monitoring (DCM version 2, software option 140)
- Workpiece touch probe
- Permitted fixture file matching the real fixture
  Further information: "Options for fixture files", Page 1172

Description of function

The **Set up fixtures** function is available as a touch probe function in the **Setup** application of the **Manual** operating mode.

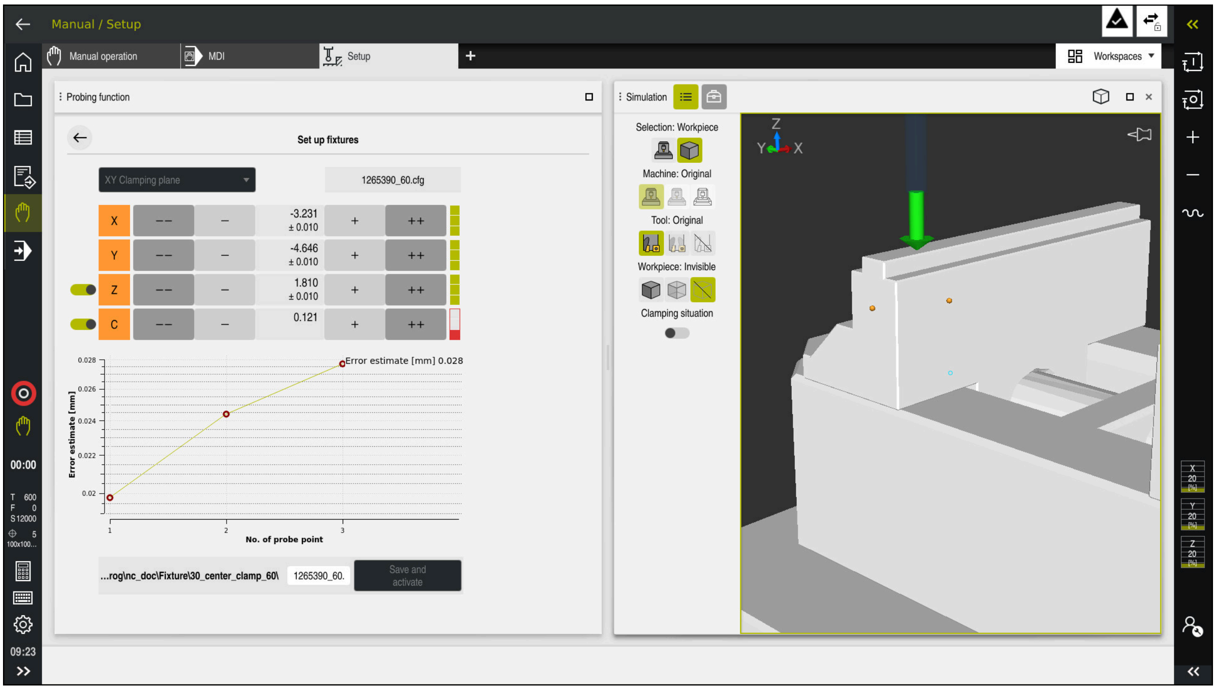
The **Set up fixtures** function determines the fixture position using various probing processes. First, one point on the fixture is probed in every linear axis. The position of the fixture is defined in this way. After probing one point in all linear axes, further points can be integrated in order to improve positioning accuracy. After defining the position in one axis direction, the control changes the status of that axis from red to green.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture for each probing point.

Further information: "Error estimate diagram", Page 1178

Extensions of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphic support for setting up the fixture.



Set up fixtures function with open **Simulation** workspace

When the **Set up fixtures** function is active, the **Simulation** workspace shows the content below:

- Current position of fixture as viewed by the control
- Probed points on the fixture
- Possible direction of probing by means of an arrow:
  - No arrow

    Probing is not possible. The workpiece touch probe is too distant from the fixture or the workpiece touch probe is positioned within the fixture, as seen by the control.

    In this case, you can adjust the position of the 3D model in the simulation, if applicable.
  - Red arrow

    Probing in the direction of the arrow is not possible.

    > Probing on edges, corners or heavily curved fixture areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    Probing in the direction of the arrow is possible under certain conditions. Probing is done in a deselected direction or might cause collisions.
  - Green arrow

    Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up fixtures** function contains the following icons and buttons:

Icon or button	Function
XY Clamping plane	This selection menu defines the plane in which the fixture is in contact with the machine. The control offers the following planes: ■ XY clamping plane ■ XZ clamping plane ■ YZ clamping plane ℹ Depending on the selected clamping plane, the control displays the corresponding axis directions. In the **XY Clamping plane**, for example, the control displays the axes **X**, **Y**, **Z** and **C**.
127_Fixture.cfg	Name of fixture file The control automatically saves the fixture file in the initial folder. The fixture file name can be edited before saving.
--	Shifts the position of the virtual fixture by 10 mm or 10° in the negative axis direction ℹ Shifts the fixture in mm in a linear axis and in degrees in a rotary axis.
-	Shifts the position of the virtual fixture by 1 mm or 1° in the negative axis direction
91.349 ± 0.007	■ Enter the position of the virtual fixture directly ■ Value and estimated accuracy after probing
+	Shifts the position of the virtual fixture by 1 mm or 1° in the positive axis direction
++	Shifts the position of the virtual fixture by 10 mm or 10° in the positive axis direction
	Status of axis The control displays the following colors: ■ Gray The axis direction is deselected for this set-up process and will not be taken into account. ■ White No probing points have been determined yet. ■ Red The control cannot determine the fixture position in this axis direction. ■ Yellow The position of the fixture in this axis direction already contains information. The information is not meaningful yet. ■ Green The control can determine the fixture position in this axis direction.

Icon or button	Function
Save and activate	This function saves all obtained data in a CFG file and activates the measured fixture in Dynamic Collision Monitoring (DCM). When using a CFG file as the data source for the measuring process, the existing CFG file can be overwritten by **Save and activate** at the end of the measuring process. When creating a new CFG file, enter a different file name next to the button.

When using a datum clamping system and for this reason you do not want to consider one axis direction (such as **Z**) when setting up the fixture, the axis in question can be deselected by a toggle switch. The control will not take deselected axis directions into account in the set-up process and positions the fixture by considering the remaining axis directions only.

Error estimate diagram

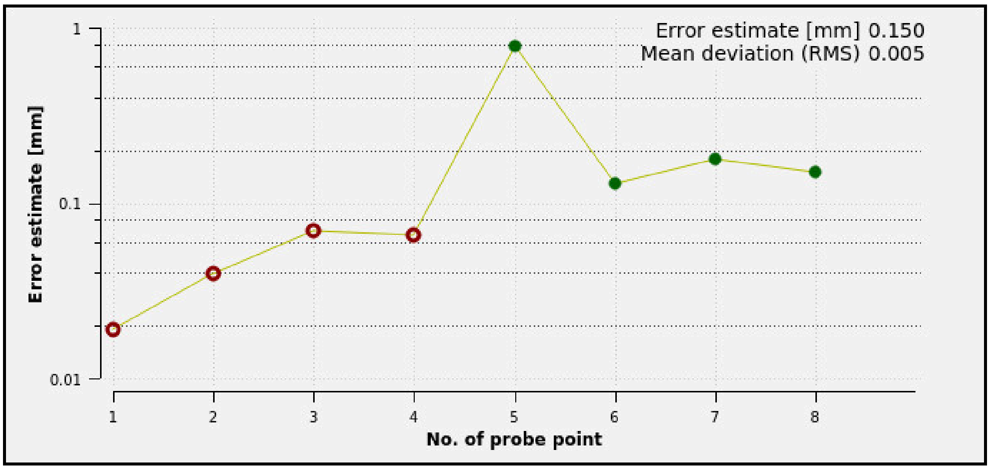
Every probing point further restricts the possible positioning of the fixture and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture. The control not only considers the probing points, but also the entire fixture.

As soon as the error estimate diagram shows green circles and the desired accuracy, the set-up process is completed.

The factors below influence the accuracy that can be achieved when measuring fixtures:

- Accuracy of workpiece touch probe
- Repeatability of workpiece touch probe
- Accuracy of 3D model
- Condition of the actual fixture (e.g., existing wear or score marks)



Error estimate diagram in the **Set up fixtures** function

The error estimate diagram of the **Set up fixtures** function displays the following information:

- **Mean deviation (RMS)**

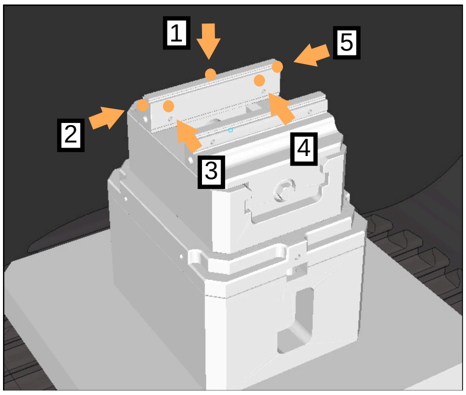
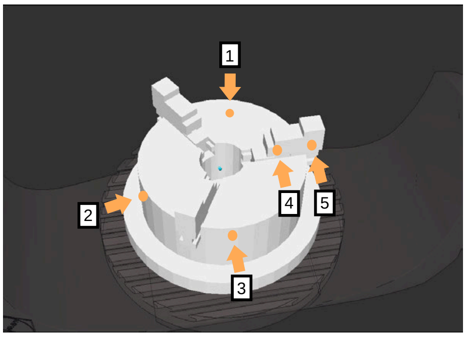
  This area shows the average distance of the measured probing points from the 3D model in mm.
- **Error estimate [mm]**

  This axis shows the course of the revised model position by means of the individual probing points. Red circles are shown until the values for all axis directions are determined. From then on, the control displays green circles.
- **No. of probe point**

  This axis shows the numbers of the individual probing points.

Example of sequence of fixture probing points

Below are some of the probing points that can be set for different fixtures:

Chucking equipment/fixtures	Possible sequence
 Probing points for a vice with a fixed vice jaw	The following probing points can be set when measuring a vice: 1 Touching the fixed vice jaw in **Z-** 2 Touching the fixed vice jaw in **X+** 3 Touching the fixed vice jaw in **Y+** 4 Touching the second value in **Y+** for rotation 5 To improve accuracy, touching the check point in **X-**
 Probing points with a three-jaw chuck	The following probing points can be set when measuring a three-point chuck: 1 Touching the jaw chuck body in **Z-** 2 Touching the jaw chuck body in **X+** 3 Touching the jaw chuck body in **Y+** 4 Touching the jaw in **Y+** for rotation 5 Touching the second value at the jaw in **Y+** for rotation

Measuring the fixed-jaw vice

> The desired 3D model must meet the requirements of the control.
> **Further information:** "Options for fixture files", Page 1172

To measure a vice using the **Set up fixtures** function:

- Affix a real vice in the working space
- Select the **Manual** operating mode
- Insert the workpiece touch probe
- Manually position the workpiece touch probe above the fixed vice jaw at a notable point

> This step makes the subsequent steps easier.

- Select the **Setup** application
- Select **Set up fixtures**
- > The control opens the **Set up fixtures** menu.
- Select a 3D model matching the real vice

Open

- Select **Open**
- > The control opens the selected 3D model in the simulation.

++

- Pre-position the 3D model by using the buttons for the individual axes within the virtual working space

> For pre-positioning the vice, use the workpiece touch probe as a point of reference.
>
> At this point in time, the control does not know the precise position of the fixture, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe and by using, for example, the table's T-slots produces values close to the position of the real vice.
>
> Even after recording the first measuring points, the shifting functions are still available for correcting the fixture position manually.

- Specify the clamping plane, e. g. **XY**
- Position the workpiece touch probe until a green down arrow appears

> As the 3D model is only pre-positioned at this point in time, the green arrow cannot provide any reliable information about whether the desired surface of the fixture will actually be touched. Check if the fixture position in the simulation and in the machine match and if touching in the direction of the arrow is possible on the machine.
>
> Do not touch directly near edges, chamfers and roundings.

- Press the **NC Start** key
- > The control probes in the direction of the arrow.
- > The control displays the status of the **Z** axis in green and shifts the fixture to the touched position. The control marks the touched position by a point in the simulation.
- Repeat this process in axis directions **X+** and **Y+**
- > The status of the axes turns green.
- Touch another point in axis direction **Y+** for the basic rotation

> To achieve maximum accuracy when touching the basic rotation, the probing points should be as far apart from one another as possible.

- > The control changes the status of the **C** axis to green.
- Touch the check point in axis direction **X-**

> Additional check points at the end of the measuring process improve the matching accuracy and minimize the faults between the 3D model and the real fixture.

Save and activate

- Select **Save and activate**
- > The control closes the **Set up fixtures** function, saves a CFG file with the measured values at the path specified above, and integrates the measured fixture into Dynamic Collision Monitoring (DCM)

Notes

NOTICE
Danger of collision! To probe the clamping situation in the machine exactly, the workpiece touch probe must be properly calibrated and the value **R2** properly defined in the tool management. Otherwise, incorrect tool data of the workpiece touch probe may cause inaccurate measurement and possibly a collision. ▶ Calibrate the workpiece touch probe at regular intervals ▶ Enter parameter **R2** in the tool management

- The control cannot identify modeling differences between the 3D model and the real fixture.
- At the time of set-up, Dynamic Collision Monitoring (DCM) does not know the exact position of the fixture. In this condition, collisions with the fixture, the tool or other non-machine components such as fixing clamps in the work envelope may occur. The non-machine components can be modeled on the control using a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 1184
- If you cancel the **Set up fixtures** function, DCM will not monitor the fixture. In this case, any fixtures previously set up are also removed from the scope of monitoring. The control displays a warning.
- Only one fixture can be measured at a time. To monitor several fixtures simultaneously by DCM, the fixtures must be integrated into a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 1184
- When measuring a jaw chuck, the coordinates of the axes **Z**, **X** and **Y** are determined just as when measuring a vice. The rotation is determined from one single jaw.
- The saved fixture file can be integrated into the NC program with the **FIXTURE SELECT** function. This can be used for simulating and executing the NC program, considering the real setup situation.

  Further information: "Loading and removing fixtures using the FIXTURE function (option 40)", Page 1183

19.2.3 Loading and removing fixtures using the FIXTURE function (option 40)

Application

The **FIXTURE** function allows loading and removing saved fixtures from within the NC program.

In the **Editor** operating mode and in the **MDI** application, different fixtures can be loaded independently of one another.

Further information: "Fixture monitoring (option 40)", Page 1171

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- A measured fixture file exists

Description of function

The selected setup situation is checked for collisions during simulation or machining.

The **FIXTURE SELECT** function selects a fixture by means of a pop-up window. The search filter in the window may have to be changed to **All files (.*)**.

The **FIXTURE RESET** function removes the fixture.

Input

11 FIXTURE SELECT "TNC:\system \Fixture\JAW_CHUCK.STL"	; Load the fixture as an STL file

The NC function includes the following syntax elements:

Syntax element	Meaning
FIXTURE	Syntax initiator for fixtures
SELECT or **RESET**	Select or remove fixture
File or **QS**	Fixture path as a fixed or variable name Only if **SELECT** has been selected

19.2.4 Editing CFG files with KinematicsDesign

Application

KinematicsDesign allows editing CFG files in the control. In this process, **KinematicsDesign** displays the fixtures graphically and thus supports troubleshooting and removal of errors. Several fixtures can be joined in order to take complex clamping situations into account in Dynamic Collision Monitoring (DCM).

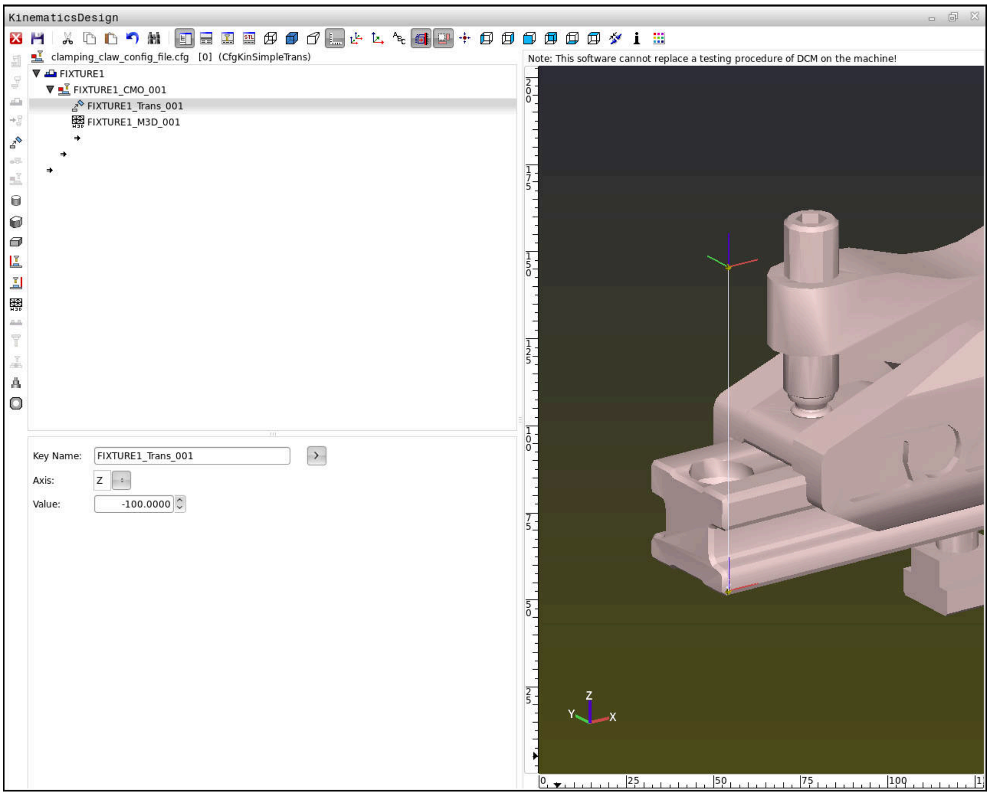
Description of function

When preparing a CFG file in the control, the control automatically opens the file with **KinematicsDesign**.

KinematicsDesign offers the following functions:

- Editing of fixtures with graphic support
- Feedback in case of incorrect entries
- Integration of transformations
- Addition of new elements
  - 3D model (M3D or STL files)
  - Cylinder
  - Prism
  - Cuboid
  - Truncated cone
  - Hole

You can integrate both STL files and M3D files into CFG files more than once.



Syntax in CFG files

The following syntax elements are used within the various CFG functions:

Function	Description
`key:= ""`	Name of the function
`dir:= ""`	Direction of a transformation (e.g., **X**)
`val:= ""`	Value
`name:= ""`	Name displayed if a collision occurs (optional input)
`filename:= ""`	File name
`vertex:= [ ]`	Position of a cube
`edgeLengths:= [ ]`	Dimensions of a cuboid
`bottomCenter:= [ ]`	Center of a cylinder
`radius:=[ ]`	Radius of a cylinder
`height:= [ ]`	Height of a geometric object
`polygonX:= [ ]`	Line of a polygon in X
`polygonY:= [ ]`	Line of a polygon in Y
`origin:= [ ]`	Starting point of a polygon

Each element is assigned its own **key**. A **key** must be unambiguous and unique, meaning that it must not occur more than once in the description of a fixture. Based on the **key**, the elements are referenced to each other.

The following functions are available if you wish to use CFG functions to describe a fixture in the control:

Function	Description
`CfgCMOMesh3D(key:="Fixture_body", filename:="1.STL",name:="")`	Definition of fixture component You can also enter an absolute path for the defined fixture component (e.g., **TNC:\nc_prog\1.STL**)
`CfgKinSimpleTrans(key:="XShiftFixture", dir:=X,val:=0)`	Shift in X axis Inserted transformations, such as a shift or rotation, are effective for all of the elements following in the kinematic chain.
`CfgKinSimpleTrans(key:="CRot0", dir:=C,val:=0)`	Rotation in C axis
`CfgCMO ( key:="fixture", primitives:= ["XShiftFixture","CRot0", "Fixture_body"], active :=TRUE, name :="")`	Describes all of the transformations contained in the fixture. The parameter active **:= TRUE** activates collision monitoring for the fixture. The **CfgCMO** contains collision objects and transformations. The fixture is combined based on the arrangement of the different transformations. Here, the transformation **XShiftFixture** shifts the center of rotation of the transformation **CRot0**.

Function	Description
`CfgKinFixModel(key:="Fix_Model", kinObjects:=["fixture"])`	Fixture designation **CfgKinFixModel** contains one or more **CfgCMO** elements.

Geometric shapes

You can add simple geometric objects to your collision object either directly in the CFG file or by using **KinematicsDesign**.

All integrated geometric shapes are subelements of the higher-order **CfgCMO**, in which they are listed as **primitives**.

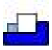
The following geometric objects are available:

Function	Description
`CfgCMOCuboid ( key:="FIXTURE_Cub", vertex:= [ 0, 0, 0 ], edgeLengths:= [0, 0, 0], name:="" )`	Definition of a cuboid
`CfgCMOCylinder ( key:="FIXTURE_Cyl", dir:=Z, bottomCenter:= [0, 0, 0], radius:=0, height:=0, name:="")`	Definition of a cylinder
`CfgCMOPrism ( key:="FIXTURE_Pris_002", height:=0, polygonX:=[], polygonY:=[], name:="", origin:= [ 0, 0, 0 ] )`	Definition of a prism A prism is described by entering the height and several polygonal lines.

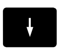
Creating a fixture entry with a collision object

The content below describes the procedure with **KinematicsDesign** opened.

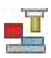
To create a fixture entry with a collision object:



- ▶ Select **Insert chucking equipment**
- > **KinematicsDesign** creates a new fixture entry within the CFG file.
- ▶ Enter a **keyname** for the fixture (e.g., **clamping jaw**)
- ▶ Confirm your input
- > **KinematicsDesign** loads the input.



- ▶ Move cursor down one level



- ▶ Select **Insert collision object**
- ▶ Confirm your input
- > **KinematicsDesign** creates a new collision object.

Defining geometric shapes

KinematicsDesign allows you to define various geometric shapes. You can construct simple fixtures by combining several geometric shapes.

To define a geometric shape:

▶ Create a fixture entry with a collision object

▶ Select the cursor key beneath the collision object

▶ Select the desired geometric shape (e.g., a cuboid)
▶ Define the position of the cuboid (e.g., **X = 0**, **Y = 0**, **Z = 0**)
▶ Define the dimensions of the cuboid (e.g., **X = 100**, **Y = 100**, **Z = 100**)
▶ Confirm your input
> The control displays the defined cuboid in the graphic.

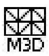
Integrating 3D models

The integrated 3D models must meet the requirements of the control.

To integrate a 3D model as a fixture:

▶ Create a fixture entry with a collision object

▶ Select the cursor key beneath the collision object



▶ Select **Insert 3D model**
> The control opens the **Open file** window.
▶ Select the desired STL or M3D file
▶ Press **OK**
> The control integrates the selected file and displays the file in the graphic window.

Fixture placement

You can place the integrated fixture at any position (e.g., for correcting the orientation of an external 3D model). For this purpose, insert transformations for all axes you wish to use.

To position a fixture with **KinematicsDesign**:

▶ Define the fixture

▶ Select the cursor key beneath the element to be positioned

▶ Select **Insert transformation**
▶ Enter a **key name** for the transformation (e.g., **Z shift**)
▶ Select the **axis** for the transformation (e.g., **Z**)
▶ Select the **value** for the transformation (e.g., **100**)
▶ Confirm your input
> **KinematicsDesign** inserts the transformation.
> **KinematicsDesign** depicts the transformation in the graphic.

Note

As an alternative to using **KinematicsDesign**, you can also create fixture files directly from the CAM system or by using the appropriate code in a text editor.

Example

The example below describes the syntax of a CFG file for a vise with two movable jaws.

Files used

Various STL files are used to describe the vise. Since the jaws of the vise are dimensionally identical, they are defined using the same STL file.

Code	Explanation
`CfgCMOMesh3D (key:="Fixture_body", filename:="vice_47155.STL", name:="")`	Body of the vise
`CfgCMOMesh3D (key:="vice_jaw_1", filename:="vice_jaw_47155.STL", name:="")`	First jaw of the vise
`CfgCMOMesh3D (key:="vice_jaw_2", filename:="vice_jaw_47155.STL", name:="")`	Second jaw of the vise

Definition of jaw opening width

In this example, the opening width of the vise is defined using two mutually dependent transformations.

Code	Explanation
`CfgKinSimpleTrans (key:="TRANS_opening_width", dir:=Y, val:=-60)`	Jaw opening width of the vise in Y direction: 60 mm
`CfgKinSimpleTrans (key:="TRANS_opening_width_2", dir:=Y, val:=30)`	Position of the first jaw of the vise in Y direction: 30 mm

Positioning of the fixture within the working space

The defined fixture components are positioned using various transformations.

Code	Explanation

`CfgKinSimpleTrans (key:="TRANS_X", dir:=X, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Y", dir:=Y, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z", dir:=Z, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z_vice_jaw", dir:=Z, val:=60)`
`CfgKinSimpleTrans (key:="TRANS_C_180", dir:=C, val:=180)`
`CfgKinSimpleTrans (key:="TRANS_SPC", dir:=C, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPB", dir:=B, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPA", dir:=A, val:=0)`
	Positioning of the fixture components
	In this example, a rotation by 180° is inserted for rotating the defined jaw of the vise. This is necessary because the same initial model is used for both jaws of the vise.
	The rotation inserted applies to all subsequent components in the transformation chain.

Description of the fixture

You need to combine all objects and transformations in the CFG file in order to ensure that the fixture is correctly depicted in the simulation.

Code	Explanation
`CfgCMO (key:="FIXTURE", primitives:= [ "TRANS_X", "TRANS_Y", "TRANS_Z", "TRANS_SPC", "TRANS_SPB", "TRANS_SPA", "Fixture_body", "TRANS_Z_vice_jaw", "TRANS_opening_width_2", "vice_jaw_1", "TRANS_opening_width", "TRANS_C_180", "vice_jaw_2" ], active:=TRUE, name:="")`	Combining the transformations and objects contained in the fixture

Fixture designation

You need to assign a designation to the combined fixture.

Code	Explanation
`CfgKinFixModel (key:="FIXTURE1", kinObjects:=["FIXTURE"])`	Designation of the combined fixture

19.3 Advanced checks in the simulation

Application

The **Advanced checks** function allows checking in the **Simulation** workspace if collisions occur between the workpiece and the tool or tool carrier.

Related topics

- Collision monitoring of machine component by Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164

Description of function

The **Advanced checks** function can be used only in the **Editor** operating mode.

The **Advanced checks** function can be activated by a toggle switch in the **Visualization options** column.

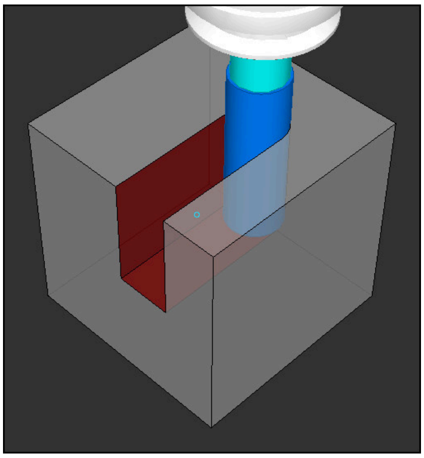
Further information: "Visualization options column", Page 1537

If the **Advanced checks** function is active, the control generates a warning in the cases below:

- Material removal at rapid traverse

  The control displays material removal at rapid traverse in red in the simulation.
- Collisions between the tool and the workpiece
- Collisions between the tool holder and the workpiece

  The control also considers inactive steps of a stepped tool.



Material removal at rapid traverse

Notes

- The **Advanced checks** function helps reduce the danger of collision. However, the control cannot consider all possible constellations during operation.
- 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: "Defining a workpiece blank with BLK FORM", Page 258

19.4 Automatic tool liftoff with FUNCTION LIFTOFF

Application

The tool retracts from the contour by up to 2 mm. The control calculates the lift off direction based on the input in the **FUNCTION LIFTOFF** block.

The **LIFTOFF** function is effective in the following situations:

- In case of an NC stop triggered by you
- In case of an NC stop triggered by the software (e.g., if an error has occurred in the drive system).
- In case of a power interruption

Related topics

- Automatic liftoff with **M148**

  Further information: "Automatically lifting off upon an NC stop or a power failure with M148", Page 1352
- Liftoff in the tool axis with **M140**

  Further information: "Retracting in the tool axis with M140", Page 1348

Requirements

- Function enabled by the machine manufacturer

  In machine parameter **on** (no. 201401), the machine manufacturer defines whether automatic lift-off is active.
- **LIFTOFF** activated for the tool

  You must define the value **Y** in the **LIFTOFF** column of the tool management.

Description of function

You can program the LIFTOFF function in the following ways:

- **FUNCTION LIFTOFF TCS X Y Z**: Lift-off in the tool coordinate system (**T-CS**) with the vector resulting from **X**, **Y** and **Z**
- **FUNCTION LIFTOFF ANGLE TCS SPB**: Lift-off in the tool coordinate system (**T-CS**) with a defined spatial angle

  This makes sense for turning (option 50)
- **FUNCTION LIFTOFF RESET**: NC function reset

Further information: "Tool coordinate system T-CS", Page 1022

The control automatically resets the **FUNCTION LIFTOFF** function at the end of a program.

FUNCTION LIFTOFF in turning mode (option 50)

NOTICE

Caution: Danger to the tool and workpiece!

Undesired movements of the axes can occur if you use the **FUNCTION LIFTOFF ANGLE TCS** function in turning mode. The behavior of the control depends on the kinematics description and Cycle **800** (**Q498 = 1**).

- Carefully test the NC program or program section in **Program run, single block** operating mode.
- If necessary, change the algebraic sign of the defined angle

If parameter **Q498** has been set to 1, the control will reverse the tool for machining.

In conjunction with the **LIFTOFF** function, the control behaves as follows:

- If the tool spindle has been defined as an axis, the **LIFTOFF** direction will be reversed.
- If the tool spindle has been defined as a kinematic transformation, the **LIFTOFF** direction will not be reversed.

Further information: "Cycle 800 ADJUST XZ SYSTEM ", Page 748

Input

11 FUNCTION LIFTOFF TCS X+0 Y+0.5 Z +0.5	; Lift off with the defined vector upon NC stop or power failure
12 FUNCTION LIFTOFF ANGLE TCS SPB +20	; Lift off with spatial angle **SPB** +20 upon NC stop or power failure

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Functions** ▶ **FUNCTION LIFTOFF**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION LIFTOFF	Syntax initiator for an automatic liftoff
TCS, **ANGLE** or **RESET**	Define the liftoff direction as a vector or a spatial angle or reset liftoff
X, **Y**, **Z**	Vector components in the tool coordinate system **T-CS** Only if **TCS** has been selected
SPB	Spatial angle in **T-CS** Only if **ANGLE** has been selected When entering 0, the control lifts off in the direction of the active tool axis.

Notes

- The control uses the **M149** function to deactivate the **FUNCTION LIFTOFF** function without resetting the lift-off direction. If you program **M148**, the control will activate the automatic lift-off of the tool in the lift-off direction defined by the **FUNCTION LIFTOFF** function.
- In case of an emergency stop, the control will not lift off the tool.
- The lift-off movement will not be monitored by Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
- In machine parameter **distance** (no. 201402), the machine manufacturer defines the maximum lift-off height.
- In the machine parameter **feed** (no. 201405), the machine manufacturer defines the speed of lift-off movement.

20

Control Functions

20.1 Adaptive Feed Control (AFC, option 45)

20.1.1 Fundamentals

Application

Adaptive Feed Control (AFC) saves time when processing NC programs and reduces wear on the machine. The control regulates the contouring feed rate during program run depending on the spindle power. In addition, the control responds to overloading of the spindle.

Related topics

- Tables related to AFC

  Further information: "Tables for AFC (option 45)", Page 2059

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Enabled by the machine manufacturer

  The machine manufacturer uses the optional machine parameter **Enable** (no. 120001) to define whether you can use AFC.

Description of function

To regulate the feed rate during program run with AFC:

- Define basic settings for AFC in the **AFC.tab** table

  Further information: "Basic AFC settings in AFC.tab", Page 2059
- Define settings for AFC for each tool in the tool management

  Further information: "Tool table tool.t", Page 1995
- Define AFC in the NC program

  Further information: "NC functions for AFC (option 45)", Page 1199
- Define AFC in the **Program Run** operating mode with the **AFC** toggle switch

  Further information: "AFC toggle switch in Program Run operating mode", Page 1201
- Prior to automatic control, determine the reference spindle power with a teach-in cut

  Further information: "AFC teach-in cut", Page 1202

If AFC is active in the teach-in cut or in control mode, the control displays an icon in the **Positions** workspace.

Further information: "Positions workspace", Page 163

Detailed information about the function is provided by the control on the **AFC** tab of the **Status** workspace.

Further information: "AFC tab (option 45)", Page 172

Benefits of AFC

Adaptive feed control (AFC) has the following advantages:

- Optimization of machining time

  By controlling the feed rate, the control tries to maintain the previously recorded maximum spindle power or the reference power specified in the tool table (**AFC-LOAD** column) during the entire machining time. It shortens the machining time by increasing the feed rate in machining zones with little material removal.
- Tool monitoring

  If the spindle power exceeds the taught-in or specified maximum value, the control reduces the feed until the reference spindle power is reached. If the minimum feed rate is exceeded, the control executes a shutdown response. AFC can also use the spindle power to monitor the tool for wear and breakage without changing the feed rate.

  Further information: "Monitoring tool wear and tool load", Page 1203
- Protection of the machine's mechanical elements

  Timely feed rate reduction and shutdown responses help to avoid machine overload.

Tables related to AFC

The control offers the following tables in conjunction with AFC:

- **AFC.tab**

  In the **AFC.TAB** table, you can enter the feed rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

  Further information: "Basic AFC settings in AFC.tab", Page 2059
- ***.H.AFC.DEP**

  With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

  Further information: "AFC.DEP settings file for teach-in cuts", Page 2063
- ***.H.AFC2.DEP**

  During a teach-in cut, the control stores information for each machining step in the **<name>.H.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you are perfoming the teach-in cut.

  In control mode, the control updates the data in this table and performs evaluations.

  Further information: "Log file AFC2.DEP", Page 2064

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

Further information: "Editing tables for AFC", Page 2066

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as Adaptive Feed Control (AFC) is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g., due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This behavior applies regardless of how the function is deactivated. This feed acceleration may result in damage to the tool and/or the workpiece!

- If the feed rate falling below the **FMIN** value is imminent, stop the machining operation without deactivating AFC
- Define the overload response for cases in which the feed rate falls below the **FMIN** value

- If adaptive feed control is active in **Control** mode, the control executes a shutdown response independent of the programmed overload response.
  - If, with the reference spindle load, the value falls below the minimum feed factor

    The control executes the shutdown response from the **OVLD** column of the **AFC.tab** table.

    Further information: "Basic AFC settings in AFC.tab", Page 2059
  - If the programmed feed rate falls below the 30% threshold

    The control executes an NC stop.
- Adaptive feed control is not intended for tools with diameters less than 5 mm. If the rated power consumption of the spindle is very high, the limit diameter of the tool may be larger.
- Do not work with adaptive feed control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.
- In NC blocks containing **FMAX**, the adaptive feed control is **not active**.
- With the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files in the file management.

20.1.2 Activating and deactivating AFC

NC functions for AFC (option 45)

Application

Adaptive Feed Control (AFC) is activated and deactivated from the NC program.

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table

  Further information: "Basic AFC settings in AFC.tab", Page 2059
- Desired control setting defined for all tools

  Further information: "Tool table tool.t", Page 1995
- **AFC** toggle switch active

  Further information: "AFC toggle switch in Program Run operating mode", Page 1201

Description of function

The control provides several functions that enable you to start and stop AFC:

- **FUNCTION AFC CTRL**: The **AFC CTRL** function activates feedback control mode starting with this NC block, even if the learning phase has not been completed yet.
- **FUNCTION AFC CUT BEGIN TIME1 DIST2 LOAD3**: The control starts a sequence of cuts with active **AFC**. The changeover from the teach-in cut to feedback control mode begins as soon as the reference power has been determined in the teach-in phase, or once one of the **TIME**, **DIST** or **LOAD** conditions has been met.
- **FUNCTION AFC CUT END**: The **AFC CUT END** function deactivates the AFC control.

Input

FUNCTION AFC CTRL

11 FUNCTION AFC CTRL	; Start AFC in control mode

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CTRL	Syntax initiator for the start of control mode

FUNCTION AFC CUT

11 FUNCTION AFC CUT BEGIN TIME10 DIST20 LOAD80	; Start AFC machining step, limit the duration of the teach-in phase

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CUT	Syntax initiator for an AFC machining step
BEGIN or **END**	Start or end machining step
TIME	End teach-in phase after the defined time in seconds Optional syntax element Only if **BEGIN** has been selected
DIST	End teach-in phase after the defined distance in mm Optional syntax element Only if **BEGIN** has been selected
LOAD	Enter the reference load of the spindle directly, max. 100% Optional syntax element Only if **BEGIN** has been selected

Notes

NOTICE

Caution: Danger to the tool and workpiece!

If you activate the **FUNCTION MODE TURN** machining mode, the control will clear the current **OVLD** values. This means that you need to program the machining mode before the tool call! If the programming sequence is not correct, no tool monitoring will take place, which might result in damage to the tool or workpiece!

- Program the **FUNCTION MODE TURN** machining mode before the tool call

- The **TIME**, **DIST** and **LOAD** defaults are modally effective. They can be reset by entering **0**.
- Execute the function **AFC CUT BEGIN** only after the starting rotational speed has been reached. If this is not the case, then the control issues an error message, and the AFC cut is not started.
- You can define a feedback-control reference power with the **AFC LOAD** tool table column and the **LOAD** input in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.

  If you program both values, the control will use the value programmed in the NC program!

AFC toggle switch in Program Run operating mode

Application

The **AFC** toggle switch allows you to activate or deactivate Adaptive Feed Control (AFC) in the **Program Run** operating mode.

Related topics

- Activating AFC in the NC program

  Further information: "NC functions for AFC (option 45)", Page 1199

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Enabled by the machine manufacturer

  The machine manufacturer uses the optional machine parameter **Enable** (no. 120001) to define whether you can use AFC.

Description of function

The **AFC** toggle switch must be activated for the NC functions for AFC to have an effect.

If you do not specifically deactivate AFC using the toggle switch, AFC remains active. The control remembers the setting of the toggle switch even if the control is restarted.

If the **AFC** toggle switch is active, the control displays an icon in the **Positions** workspace. In addition to the current setting of the feed rate potentiometer, the control shows the controlled feed value as a percentage (%).

Further information: "Positions workspace", Page 163

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as the AFC function is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g. due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This applies regardless of how the function is deactivated (e.g. feed rate potentiometer). This acceleration may result in damages to the tool or the workpiece!

- If it is imminent that the feed rate falls below the **FMIN** value, stop the machining operation (instead of deactivating the **AFC** function)
- Define the overload response for cases in which the feed rate falls below the **FMIN** value

- If Adaptive Feed Control is active in **Control** mode, the control internally sets the spindle override to 100%. This means that you can no longer change the spindle speed.
- If adaptive feed control is active in **Control** mode, the control loads the value from the feed-rate override function.
  - Increasing the feed-rate override has no influence on the control.
  - If you reduce the feed override with the potentiometer by more than 10% in relation to the position at the start of the program, the control switches AFC off.

    You can reactivate control with the **AFC** toggle switch.
  - Potentiometer values of up to 50% always have an effect, even with active control.
- Mid-program startup is allowed during active feed control. The control takes the cutting number of the startup block in account.

20.1.3 AFC teach-in cut

Application

With the teach-in cut, the control determines the reference power of the spindle for the machining step. Based on the reference power, the control adjusts the feed rate in control mode.

If you have already determined the reference power for a machining operation, you can specify the value for the machining operation. For this, the control provides the **AFC-LOAD** column in the tool management and the **LOAD** syntax element in the **FUNCTION AFC CUT BEGIN** function. In this case, the control no longer performs a teach-in cut, but uses the specified value immediately for control.

Related topics

- Enter the known reference power in the **AFC-LOAD** column in the tool management

  Further information: "Tool table tool.t", Page 1995
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function

  Further information: "NC functions for AFC (option 45)", Page 1199

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table

  Further information: "Basic AFC settings in AFC.tab", Page 2059
- Desired control setting defined for all tools

  Further information: "Tool table tool.t", Page 1995
- Desired NC program selected in the **Program Run** operating mode
- **AFC** toggle switch active

  Further information: "AFC toggle switch in Program Run operating mode", Page 1201

Description of function

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**.

Further information: "AFC.DEP settings file for teach-in cuts", Page 2063

When you are performing a teach-in cut, the control shows the spindle reference power determined until this time in a pop-up window.

When the control has determined the control reference power, it ends the teach-in cut and switches to control mode.

Notes

- When you record a teach-in cut, the control internally sets the spindle override to 100%. Then you can no longer change the spindle speed.
- During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.
- You can repeat a teach-in cut as often as desired. Manually change the status from **ST** back to **L**. If the programmed feed rate value is far too high and forces you to sharply decrease the feed rate override during the machining step, you will have to repeat the teach-in cut.
- If the determined reference load is greater than 2%, the control changes the status from teach-in (**L**) to controlling (**C**). Adaptive feed control is not possible for smaller values.
- In **FUNCTION MODE TURN** machining mode, the minimum reference load is 5%. Even if the control determines lower values, it will still use this minimum reference load. Thus, the overload limits (indicated as percentage values) are based on a minimum reference load of at least 5%.

20.1.4 Monitoring tool wear and tool load

Application

With Adaptive Feed Control (AFC), you can monitor the tool for wear and breakage. The **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management can be used for this.

Related topics

- **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management

  Further information: "Tool table tool.t", Page 1995

Description of function

If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active.

Further information: "Basic AFC settings in AFC.tab", Page 2059

Tool wear monitoring

Activate cut-related tool wear monitoring by entering a value not equal to 0 in the **AFC-OVLD1** column in the tool table.

The overload response depends on the **AFC.TAB** column **OVLD**.

In conjunction with cut-related tool wear monitoring, the control only evaluates the options **M**, **E**, and **L** in the **OVLD** column. The following responses are possible:

- Pop-up window
- Lock current tool
- Insert replacement tool

Tool load monitoring

Activate cut-related tool load monitoring (tool breakage control) by entering a value not equal to 0 in the **AFC-OVLD2** column in the tool table.

As overload response, the control always executes a machining stop and locks the momentary tool.

In turning mode, the control can check for tool wear and tool breakage.

A tool breakage leads to a sudden load decrease. If you want the control to monitor the load decrease, too, enter the value 1 in the SENS column.

Further information: "Basic AFC settings in AFC.tab", Page 2059

20.2 Active Chatter Control (ACC, option 145)

Application

Chatter marks can be caused during heavy-duty machining, in particular. **ACC** reduces chattering, thereby reducing wear on the tool and machine. In addition, **ACC** increases metal removal rates.

Related topics

- **ACC** column in the tool table

  Further information: "Tool table tool.t", Page 1995

Requirements

- Active Chatter Control (ACC, software option 145)
- Control adapted by the machine manufacturer
- **ACC** column in the tool management defined with **Y**
- Number of tool cutting edges defined in the **CUT** column

Description of function

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the machine can sometimes begin to **chatter**. This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

In order to reduce a machine's tendency to chatter, HEIDENHAIN offers an effective control function known as Active Chatter Control (**ACC**). The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. Depending on the type of machine, the metal-removal rate can often be increased by more than 25%. You reduce the mechanical load on the machine and increase the life of your tools at the same time.

ACC was developed especially for roughing and heavy machining and is particularly effective in this area. You need to conduct appropriate tests to see whether ACC will also be advantageous on your machine and with your tool.

ACC is activated and deactivated using the **ACC** toggle switch in the **Program Run** operating mode or the **MDI** application.

Further information: "Program Run operating mode", Page 1954

Further information: "The MDI Application ", Page 1933

If ACC is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 163

Notes

- ACC reduces or prevents vibrations in the range of 20 Hz to 150 Hz. If ACC does not appear to have an effect, the vibrations may be outside of this range.
- With Machine Vibration Control (MVC, software option 146), you can also positively influence the result.

20.3 Functions for controlling program run

20.3.1 Overview

The control provides the following NC functions for program control:

Syntax	Function	Further information
FUNCTION S-PULSE	Program pulsing spindle speed	Page 1206
FUNCTION DWELL	Program singular dwell time	Page 1207
FUNCTION FEED DWELL	Program cyclic dwell time	Page 1208

20.3.2 Pulsing spindle speed with FUNCTION S-PULSE

Application

Using the **S-PULSE FUNCTION** you can program a pulsing spindle speed (e.g., to avoid natural oscillations of the machine) when operating at a constant spindle speed.

Description of function

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 effective. Both input values are optional. If you do not define a parameter, the function applies to the entire speed range.

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

When a pulsing spindle speed is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 163

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 effective Only if **P-TIME** has been selected Optional syntax element
TO-SPEED	Upper speed limit up to which the pulsing spindle speed will be effective Only if **P-TIME** has been selected Optional syntax element

Note

The control never exceeds a programmed speed limit. The spindle speed is maintained until the sinusoidal curve of the **S-PULSE FUNCTION** falls below the maximum speed once more.

20.3.3 Programmed dwell time with FUNCTION DWELL

Application

The **FUNCTION DWELL** function enables you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

Related topics

- Cycle **9 DWELL TIME**

  Further information: "Cycle 9 DWELL TIME ", Page 1209
- Program recurring dwell time

  Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 1208

Description of function

The defined dwell time from **FUNCTION DWELL** is effective in both milling and turning operations.

Input

11 FUNCTION DWELL TIME10	; Dwell time for 10 seconds
12 FUNCTION DWELL REV5.8	; Dwell time for 5.8 spindle revolutions

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION DWELL	Syntax initiator for singular dwell time
TIME or **REV**	Duration of dwell time in seconds or spindle revolutions

20.3.4 Cyclic dwell time with FUNCTION FEED DWELL

Application

The **FUNCTION FEED DWELL** function can be used to program a cyclic dwell time in seconds (e.g., to force chip breaking in a turning cycle).

Related topics

- Program a one-time dwell time
  Further information: "Programmed dwell time with FUNCTION DWELL", Page 1207

Description of function

The defined dwell time from **FUNCTION FEED DWELL** is effective in both milling and turning operations.

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motion.

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

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

Program **FUNCTION FEED DWELL** immediately prior to the operation you wish to run with chip breaking. Reset the dwell time immediately following the machining with chip breaking.

Input

11 FUNCTION FEED DWELL D-TIME0.5 F-TIME5	; Activate cyclic dwell time: Machine for 5 seconds, dwell for 0.5 seconds

To navigate to this function:

Insert NC function ▶ **Special functions** ▶ **Functions** ▶ **FUNCTION FEED** ▶ **FUNCTION FEED DWELL**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FEED DWELL	Syntax initiator for cyclic dwell time
D-TIME or **RESET**	Define dwell time duration in seconds or reset recurring dwell time
F-TIME	Duration of machining time until the next dwell time in seconds Only if **D-TIME** is selected

Notes

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

- You can also reset the dwell time by entering **D-TIME 0**.

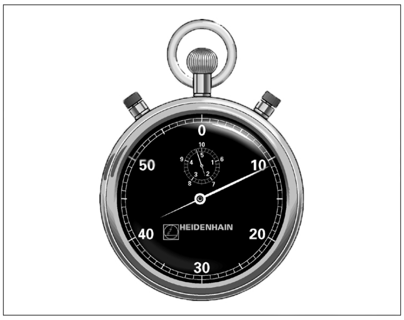
20.4 Cycles with control function

20.4.1 Cycle 9 DWELL TIME

ISO programming
G4

Application

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.



Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.

The cycle becomes effective as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Related topics

- Dwell time with **FUNCTION FEED DWELL**

  Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 1208
- Dwell time with **FUNCTION DWELL**

  Further information: "Programmed dwell time with FUNCTION DWELL", Page 1207

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

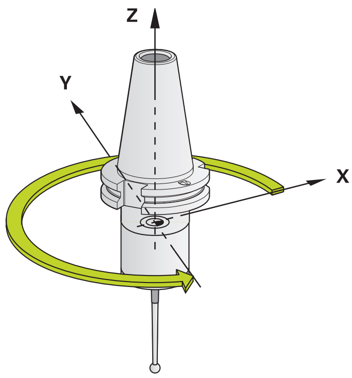
```
89 CYCL DEF 9.0  DWELL TIME
90 CYCL DEF 9.1  DWELL 1.5
```

20.4.2 Cycle 13 ORIENTATION

ISO programming
G36

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**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **13** is used internally for Cycles **202**, **204**, and **209**. Please note that, if required, you must program Cycle **13** again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

Help graphic	Parameter
	Orientation angle Enter the angle relative to the angle reference axis of the working plane. Input: **0...360**

Example

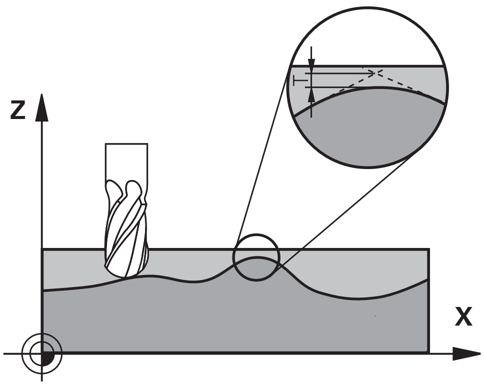
```
11 CYCL DEF 13.0 ORIENTATION
12 CYCL DEF 13.1 ANGLE180
```

20.4.3 Cycle 32 TOLERANCE

ISO programming
G62

Application

Refer to your machine manual.
Machine and control must be specially prepared by the machine manufacturer for use of this cycle.



With the entries in Cycle **32** you can influence the result of HSC machining with respect to accuracy, surface definition and speed, in as much as the control has been adapted to the machine's characteristics.

The control automatically smooths the contour between any two contour elements (whether compensated or not). This means that the tool has constant contact with the workpiece surface and therefore reduces wear on the machine tool. The tolerance defined in the cycle also affects the traverse paths on circular arcs.

If necessary, the control automatically reduces the programmed feed rate so that the program can be executed at the fastest possible speed without jerking. **Even if the control does not move the axes with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the control can move the axes.

Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error (**tolerance value**) is set in a machine parameter by the machine manufacturer. With **Cycle 32** you can change the pre-set tolerance value and select different filter settings, provided that your machine manufacturer has implemented these features.

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the control, but by the fact that, in order to machine the contour transitions very exactly, the control might have to drastically reduce the speed.

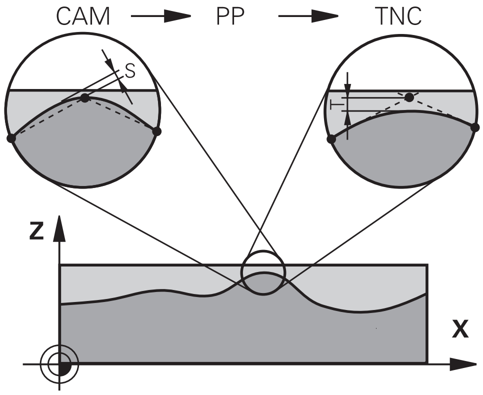
Resetting

The control resets Cycle **32** if you do one of the following:

- Redefine Cycle **32** and confirm the dialog prompt for the **tolerance value** with **NO ENT**
- Select a new NC program

After you have reset Cycle **32**, the control reactivates the tolerance that was predefined by the machine parameters.

Influences of the geometry definition in the CAM system



The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system. The chord error defines the maximum point spacing of NC programs generated in a postprocessor (PP). If the chord error is less than or equal to the tolerance value **T** defined in Cycle **32**, then the control can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing of the contour if you choose a tolerance value in Cycle **32** between 110 % and 200 % of the CAM chord error.

Related topics

- Working with CAM-generated NC programs

  Further information: "CAM-generated NC programs", Page 1301

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **32** is DEF-active which means that it becomes effective as soon as it is defined in the NC program.
- In a program with millimeters set as unit of measure, the control interprets the entered tolerance value **T** in millimeters. In an inch program it interprets it as inches.
- If you load an NC program with Cycle **32** that contains only the **Tolerance value** T cycle parameter, the control inserts the two remaining parameters with the value 0 if required.
- As the tolerance value increases, the diameter of circular movements usually decreases, unless HSC filters are active on your machine (set by the machine manufacturer).
- If Cycle **32** is active, the control shows the defined cycle parameters on the **CYC** tab of the additional status display.

Keep the following in mind for 5-axis simultaneous machining!

- NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more uniform. In Cycle **32**, you can additionally set a higher rotary axis tolerance **TA** (e.g., between 1° and 3°) for an even more constant feed-rate curve at the tool center point (TCP).
- For NC programs for 5-axis simultaneous machining with toroid cutters or spherical cutters, where the NC output is for the south pole of the sphere, choose a lower rotary axis tolerance. 0.1° is a typical value. However, the maximum permissible contour damage is the decisive factor for the rotary axis tolerance. This contour damage depends on the possible tool tilting, tool radius and engagement depth of the tool.
  With 5-axis hobbing with an end mill, you can calculate the maximum possible contour damage T directly from the cutter engagement length L and permissible contour tolerance TA:
  T ~ K x L x TA K = 0.0175 [1/°]
  Example: L = 10 mm, TA = 0.1°: T = 0.0175 mm

Sample formula for a toroid cutter:

When machining with a toroid cutter, the angle tolerance is very important.

$$Tw = \frac{180}{\pi * R} T_{32}$$

T_w: Angle tolerance in degrees
π: Circular constant (pi)
R: Major radius of the torus in mm
T_{32}: Machining tolerance in mm

Cycle parameters

Help graphic	Parameter
	Tolerance value T Permissible contour deviation in mm (or inches with inch programming) **> 0**: If you enter a value greater than zero, the control will use the maximum permissible deviation you have specified. **0**: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine manufacturer Input: **0...10**
	HSC mode, finishing = 0, roughing = 1 Activate filter: **0**: Milling with increased contour accuracy. The control uses internally defined finishing filter settings. **1**: Milling with increased feed rate. The control uses internally defined roughing filter settings. Input: **0**, **1**
	Tolerance TA for rotary axes Permissible position error of rotary axes in degrees with active **M128** (**FUNCTION TCPM**). The control always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for NC programs for more than one axis by entering a large tolerance value (e.g., 10°), because the control does not always have to position the rotary axis exactly at the given nominal position. The tool orientation (position of the rotary axis with respect to the workpiece surface) will be adjusted. The position at the **T**ool **C**enter **P**oint (**TCP**) will be corrected automatically. For example, with a spherical cutter measured in its center and programmed based on the center path, there will be no adverse effects on the contour. **> 0**: If you enter a value greater than zero, the control will use the maximum permissible deviation you have specified. **0**: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine manufacturer. Input: **0...10**

Example

```
11 CYCL DEF 32.0 TOLERANCE
12 CYCL DEF 32.1 T0.05
13 CYCL DEF 32.2 HSC-MODE:1 TA5
```

20.5 Global Program Settings (GPS, option 44)

20.5.1 Fundamentals

Application

The Global Program Settings (GPS) allow you to define selected transformations and settings without changing the NC program. All of the settings apply globally and are superimposed on the relevant active NC program.

Related topics

- Coordinate transformations in the NC program

  Further information: "NC functions for coordinate transformation", Page 1045

  Further information: "Coordinate transformation cycles", Page 1034
- **GPS** tab in the **Status** workspace.

  Further information: "GPS tab (option 44)", Page 174
- Reference systems of the control

  Further information: "Reference systems", Page 1010

Requirement

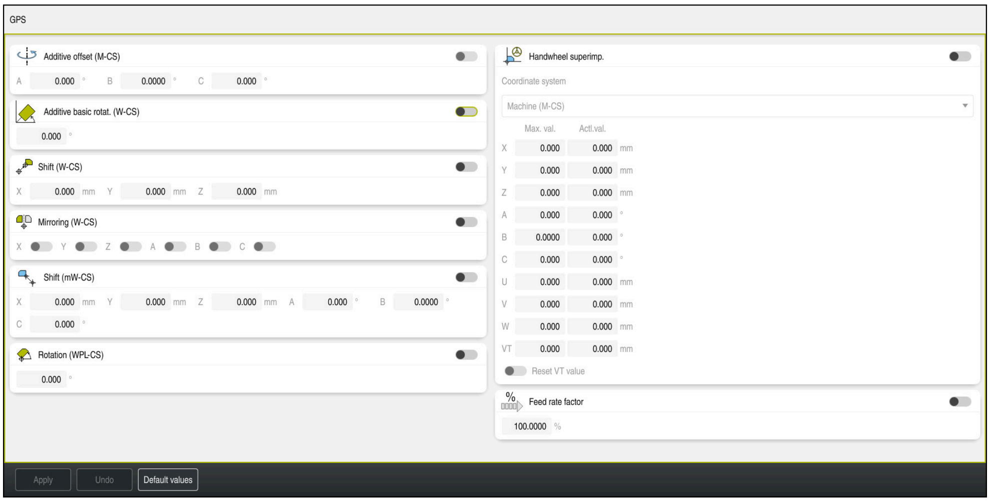
- Global Program Settings (GPS, software option 44)

Description of function

The values of the Global Program Settings are defined and activated in the **GS** workspace.

The **GS** workspace is available in the **Program Run** operating mode and in the **MDI** application of the **Manual** operating mode.

The transformations of the **GS** workspace are effective in all operating modes and are persistent across reboots of the control.



GS workspace with active functions

The functions of GPS are activated using toggle switches.

The control marks the sequence in which the transformations are effective with green digits.

The control shows the active settings of GPS on the **GPS** tab of the **Status** workspace.

Further information: "GPS tab (option 44)", Page 174

Before executing an NC program with active GPS in the **Program Run** operating mode, you must confirm use of the GPS functions in a pop-up window.

Buttons

The control provides the following buttons in the **GS** workspace:

Button	Description
Apply	Save changes in the **GS** workspace
Undo	Reset unsaved changes in the **GS** workspace
Default values	Set the **Feed rate factor** function to 100%, reset all other functions to zero

Overview of Global Program Settings (GPS)

The Global Program Settings (GPS) include the following functions:

Function	Description
Additive offset (M-CS)	Shift of the zero position of an axis in the machine coordinate system **M-CS** **Further information:** "Function Additive offset (M-CS)", Page 1219
Additive basic rotat. (W-CS)	Additional rotation based on basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. **Further information:** "Function Additive basic rotat. (W-CS)", Page 1222
Shift (W-CS)	Shift of workpiece preset in a single axis in the workpiece coordinate system **W-CS** **Further information:** "Function Shift (W-CS)", Page 1222
Mirroring (W-CS)	Mirroring of individual axes in the workpiece coordinate system **W-CS** **Further information:** "Function Mirroring (W-CS)", Page 1223
Shift (mW-CS)	Additional shift of a workpiece datum already shifted in the modified workpiece coordinate system **(mW-CS)**. **Further information:** "Function Shift (mW-CS)", Page 1224
Rotation (WPL-CS)	Rotation around the active tool axis in the working plane coordinate system **WPL-CS** **Further information:** "Function Rotation (WPL-CS)", Page 1225
Handwheel superimposition	Superimposed movement of NC program positions with the electronic handwheel **Further information:** "Function Handwheel superimp.", Page 1225
Feed rate factor	Manipulation of the active feed rate **Further information:** "Function Feed rate factor", Page 1228

Define and activate Global Program Settings (GPS)

To define and activate the Global Program Settings (GPS):

- Select an operating mode (e.g., **Program run**)
- Open the **GS** workspace
- Activate the toggle switch for the required function (e.g., **Additive offset (M-CS)**)
- > The control activates the selected function.
- Enter a value in the desired field (e.g., **A=10.0°**)

Apply

- Press **Apply**
- > The control accepts the entered values.

> If you select an NC program for program run, you must confirm the Global Program Settings (GPS).

Resetting Global Program Settings (GPS)

To reset the Global Program Settings (GPS):

- Select an operating mode (e.g., **Program Run**)
- Open the **GS** workspace

Default values

- Select **Default values**

> Provided that you have not selected the **Apply** button, you can restore the values with the **Undo** function.

- > The control sets the values of all Global Program Settings (GPS) to zero except for the feed factor.
- > The control sets the feed factor to 100%.

Apply

- Press **Apply**
- > The control saves the values that have been reset.

Notes

- The control grays out any axes that are not active on your machine.
- Value inputs are defined in the selected unit of measurement for the position display (mm or inch). These values include offset values and values of **Handwheel superimp.** Angles are always entered in degrees.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) temporarily.
- The optional machine parameter **CfgGlobalSettings** (no. 128700) can be used to define which GPS functions are available on the control. The machine manufacturer enables this parameter.

20.5.2 Function Additive offset (M-CS)

Application

With the **Additive offset (M-CS)** function, you can shift the zero position of a machine axis in the machine coordinate system **M-CS**. You can use this function, for example, on large machines, to compensate an axis when using axis angles.

Related topics

- Machine coordinate system **M-CS**
  Further information: "Machine coordinate system M-CS", Page 1012
- Difference between basic rotation and offset
  Further information: "Basic transformation and offset", Page 2039

Description of function

The control adds the value to the active axis-specific offset from the preset table.

Further information: "Preset table", Page 2035

If you activate a value in the **Additive offset (M-CS)** function, the zero position of the affected axis changes in the position display of the **Positions** workspace. The control assumes a different zero position of the axes.

Further information: "Positions workspace", Page 163

Application example

The travel range of a machine with AC fork head is increased using the **Additive offset (M-CS)** function. An eccentric tool chuck is used and the zero position of the C axis is shifted by 180°.

Initial situation:

- Machine kinematics with AC fork head
- Use of an eccentric tool chuck
  The tool is clamped in an eccentric tool chuck outside the center of rotation of the C axis.
- The machine parameter **presetToAlignAxis** (no. 300203) for the C axis is set to **FALSE**

To increase the traversing distance:

- ▶ Open the **GS** workspace
- ▶ Activate the **Additive offset (M-CS)** toggle switch
- ▶ Enter **C 180°**

- ▶ Press **Apply**
- ▶ Program a positioning movement with **L C+0** in the desired NC program
- ▶ Select an NC program
- > The control considers the 180° rotation for all C axis positioning movements as well as the changed tool position.
- > The position of the C axis does not affect the position of the workpiece preset.

Notes

- After having activated an additive offset, reset the workpiece preset.
- In the optional machine parameter **presetToAlignAxis** (no. 300203), the machine manufacturer defines for each axis how the control interprets offsets for the following NC functions:
  - **FUNCTION PARAXCOMP**

    Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1284
  - **FUNCTION POLARKIN** (option 8)

    Further information: "Machining with polar kinematics with FUNCTION POLARKIN", Page 1295
  - **FUNCTION TCPM** or **M128** (option 9)

    Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
  - **FACING HEAD POS** (option 50)

    Further information: "Using a facing slide with FACING HEAD POS (option 50)", Page 1291

20.5.3 Function Additive basic rotat. (W-CS)

Application

One possible use for the **Additive basic rotat. (W-CS)** function is better of the working space. For example, you can rotate an NC program by 90° so that the X and Y directions are inverted during execution.

Description of function

The function **Additive basic rotat. (W-CS)** takes effect in addition to the basic rotation or 3D basic rotation from the preset table. The values of the preset table do not change in this respect.

Further information: "Preset table", Page 2035

The function **Additive basic rotat. (W-CS)** has no effect on the position display.

Application example

You rotate the CAM output of an NC program by 90° and compensate for the rotation using the function **Additive basic rotat. (W-CS)**.

Initial situation:

- Available CAM output for gantry-type milling machine with a large range of traverse of the Y axis
- The available machining center has the necessary traversing range only in the X axis
- The workpiece blank is clamped with a 90° rotation (long side along the X axis)
- The NC program must be rotated by 90° (algebraic sign depends on the preset position)

To rotate the CAM output:

- Open the **GS** workspace
- Activate the toggle switch for **Additive basic rotat. (W-CS)**
- Enter **90°**

- Press **Apply**
- Select NC program
- > The control considers the 90° rotation for all axis positioning movements.

20.5.4 Function Shift (W-CS)

Application

You can use the **Shift (W-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult, for example.

Description of function

The **Shift (W-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 1016

The **Shift (W-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 188

Application example

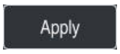
The surface of a workpiece to be reworked is determined using the handwheel and the offset is compensated for using the **Shift (W-CS)** function.

Initial situation:

- Reworking of a free-form surface is required
- Workpiece clamped
- Basic rotation and workpiece preset measured in the working plane
- Z coordinate must be defined with the handwheel due to the presence of a free-form surface

To shift the workpiece surface of a workpiece to be reworked:

- Open the **GS** workspace
- Activate the **Handwheel superimp.** toggle switch
- Determine the workpiece surface by scratching, using the handwheel
- Activate the **Shift (W-CS)** toggle switch
- Transfer the determined value to the corresponding axis of the **Shift (W-CS)** function (e.g. **Z**)



- Press **Apply**
- Start an NC program
- Activate **Handwheel superimp.** with the **Workpiece (WPL-CS)** coordinate system
- Determine the workpiece surface by scratching, using the handwheel for fine adjustment
- Select NC program
- > The control takes the **Shift (W-CS)** into account.
- > The control uses the current values from **Handwheel superimp.** in the **Workpiece (WPL-CS)** coordinate system.

20.5.5 Function Mirroring (W-CS)

Application

You can use the **Mirroring (W-CS)** function to execute mirror-inverted execution of an NC program without having to modify the NC program.

Description of function

The **Mirroring (W-CS)** function acts on an axis-by-axis basis. The value is additive to mirroring defined in the NC program before tilting the working plane with Cycle **8 MIRRORING** or the **TRANS MIRROR** function.

Further information: "Cycle 8 MIRRORING", Page 1036

Further information: "Mirroring with TRANS MIRROR", Page 1047

The **Mirroring (W-CS)** function has no effect on the position display in the **Positions** workspace.

Further information: "Position displays", Page 188

Application example

An NC program can be edited with the **Mirroring (W-CS)** function mirror-inverted.

Initial situation:

- Available CAM output for right mirror cap
- NC program set to the center of the ball-nose cutter and **FUNCTION TCPM** function with spatial angles output
- The workpiece datum is centered on the workpiece blank
- Mirroring required in the X axis to produce the left mirror cap

To mirror the CAM output of an NC program:

- Open the **GS** workspace
- Activate the **Mirroring (W-CS)** toggle switch
- Activate the **X** toggle switch

- Press **Apply**
- Run the NC program
- > The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.

Notes

- If you use **PLANE** functions or the **FUNCTION TCPM** function with spatial angles, the rotary axes are mirrored accordingly along with the mirrored main axes. This always creates the same constellation, regardless of whether the rotary axes were marked in the **GS** workspace.
- With **PLANE AXIAL**, the mirroring of rotary axes is irrelevant.
- With the **FUNCTION TCPM** function with axis angles, you must activate all axes to be mirrored individually in the **GS** workspace.

20.5.6 Function Shift (mW-CS)

Application

You can use the **Shift (mW-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult in the modified workpiece coordinate system **mW-CS**, for example.

Description of function

The **Shift (mW-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 1016

The **Shift (mW-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 188

A modified workpiece coordinate system **mW-CS** is present with active **Shift (W-CS)** or active **Mirroring (W-CS)**. Without these preceding coordinate transformations, the **Shift (mW-CS)** option would be effective directly in the workpiece coordinate system (**W-CS**) and would thus be identical to **Shift (W-CS)**.

Application example

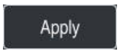
You mirror the CAM output of an NC program. After mirroring, you shift the workpiece datum in the mirrored coordinate system to produce the counterpart to a mirror cap.

Initial situation:

- Available CAM output for right mirror cap
- The workpiece datum is located in the left front corner of the workpiece blank.
- NC program set to the center of the ball-nose cutter and **Function TCPM** function with spatial angles output
- The left mirror cap is to be machined

To shift the datum in the mirrored coordinate system:

- Open the **GS** workspace
- Activate the **Mirroring (W-CS)** toggle switch
- Activate the **X** toggle switch
- Activate the **Shift (mW-CS)** toggle switch
- Enter the value for shifting the workpiece datum in the mirrored coordinate system



- Press **Apply**
- Run the NC program
- > The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.
- > The control takes the modified position of the workpiece datum into account.

20.5.7 Function Rotation (WPL-CS)

Application

With the **Rotation (WPL-CS)** function, you can, for example, compensate for the misalignment of a workpiece in the already swiveled working plane coordinate system **WPL-CS** without modifying the NC program.

Description of function

The **Rotation (WPL-CS)** function is active in the tilted working plane coordinate system **WPL-CS**. The value is added to a rotation in the NC program with Cycle **10 ROTATION** or the **TRANS ROTATION** function.

Further information: "Rotations with TRANS ROTATION", Page 1050

The **Rotation (WPL-CS)** function has no effect on the position display.

20.5.8 Function Handwheel superimp.

Application

With the **Handwheel superimp.** function, you can traverse the axes with the superimposed handwheel during program run. You select the coordinate system in which the **Handwheel superimp.** function is effective.

Related topics

- Handwheel superimpositioning with **M118**

  Further information: "Activating handwheel superimpositioning with M118", Page 1334

Description of function

In the **Max. val.** column, you define the maximum traversing distance for the respective axis. The traverse can be either in the positive or in the negative direction. The maximum path is therefore twice as large as the input value.

In the **Actl.val.** column, the control displays the path traversed using the handwheel for each axis.

The **Actl.val.** column can also be edited manually. If you enter a value greater than the **Max. val.**, you cannot activate the value. The control marks an incorrect value in red. The control displays a warning message and prevents the form from being closed.

If the **Actl.val.** column contains a value when you activate the function, the control will use the menu for returning to move to the new position.

Further information: "Returning to the contour", Page 1972

The **Handwheel superimp.** function affects the position display in the **Positions** workspace. The control shows the values offset by the handwheel in the position display.

Further information: "Positions workspace", Page 163

The control displays the values of the two methods for **Handwheel superimp.** on the **POS HR** tab of the additional status display.

On the **POS HR** tab of the **Status** workspace, the control shows whether the **Max. val.** is defined using the **M118** function or the Global Program Settings (GPS).

Further information: "POS HR tab", Page 180

Virtual tool axis VT

The virtual tool axis **VT** is needed for machining operations with inclined tools (e.g., for manufacturing oblique holes without using a tilted working plane).

Handwheel superimp. can also be executed in the active tool axis direction. The **VT** always corresponds to the direction of the active tool axis. On machines with head rotation axes, this direction may not correspond to the basic coordinate system **B-CS**. You activate the function with the **VT** line.

Further information: "Notes concerning different machine kinematics", Page 1054

By default, values traversed with the handwheel in the **VT** remain active even after a tool change. If you activate the **Reset VT value** toggle switch, the control resets the actual value of the **VT** when a tool is changed.

The control displays the values of the virtual tool axis **VT** on the **POS HR** tab of the **Status** workspace.

Further information: "POS HR tab", Page 180

For the control to display values, you must define a value greater than 0 in the **VT** function for **Handwheel superimp.**

Notes

NOTICE

Danger of collision!

The coordinate system chosen in the selection menu also takes effect on **Handwheel superimp.** with **M118**, even if the Global Program Settings function (GPS) is not active. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- Before exiting the form, always make sure to select the **Machine (M-CS)** coordinate system
- Test the behavior at the machine

NOTICE

Danger of collision!

When both methods for **Handwheel superimp.** with **M118** and with the Global Program Settings GPS are active at the same time, the definitions influence each other, depending on their sequence of activation. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- Use only one method for **Handwheel superimp.**
- Preferably use the **Handwheel superimp.** option of the **Global Program Settings** function
- Test the behavior at the machine

HEIDENHAIN does not recommend using both methods for **Handwheel superimp.** at the same time. If **M118** cannot be removed from the NC program, you should at least activate **Handwheel superimp.** from GPS prior to selecting the program. This ensures that the control uses the GPS function rather than **M118**.

- If neither the NC program nor the Global Program Settings were used to activate coordinate system transformations, **Handwheel superimp.** is effective in the same manner in all coordinate systems.
- If you want to use **Handwheel superimp.** while machining with active Dynamic Collision Monitoring (DCM), then the control must be in a stopped or interrupted state. Alternatively, you can also deactivate DCM.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
- **Handwheel superimp.** in virtual axis direction **VT** requires neither a **PLANE** function nor the **FUNCTION TCPM** function.
- Use the machine parameter **axisDisplay** (no. 100810) to define whether the control also shows the virtual axis **VT** in the position display of the **Positions** workspace.

  Further information: "Positions workspace", Page 163

20.5.9 Function Feed rate factor

Application

You can use the **Feed rate factor** function to influence the effective feed rates on the machine (e.g. to adjust the feed rates of a CAM program). This will prevent the CAM program from being re-output using the postprocessor. When doing so, you change all feed rates as a percentage without making any changes in the NC program.

Related topics

- Feed rate limit **F MAX**

  The **Feed rate factor** function has no influence on the feed rate limit with **F MAX**.

  Further information: "Feed rate limit FMAX", Page 1958

Description of function

All feed rates are changed as a percentage. You define a percentage value from 1% to 1000%.

The **Feed rate factor** function acts on the programmed feed rate and the feed rate potentiometer, but not on rapid traverse **FMAX**.

The control shows the current feed rate in field **F** of the **Positions** workspace. If the **Feed rate factor** function is active, the feed rate is shown with the defined values taken into account.

Further information: "Presets and technology values", Page 165

21

Monitoring

21.1 Component Monitoring with MONITORING HEATMAP (option 155)

Application

The **MONITORING HEATMAP** function allows you to start and stop the workpiece representation in a component heatmap from within the NC program.

The control monitors the selected component and shows the result in a color-coded heatmap on the workpiece.

> If the process monitoring (option 168) in the simulation displays a process heat map, the control does not display a component heat map.
>
> **Further information:** "Process Monitoring (option 168)", Page 1236

Related topics

- **MON** tab in the **Status** workspace

  Further information: "MON tab (option 155)", Page 177
- Cycle **238 MEASURE MACHINE STATUS** (option 155)

  Further information: "Cycle 238 MEASURE MACHINE STATUS (option 155)", Page 1233
- Color the workpiece as a heat map in the simulation

  Further information: "Workpiece options column", Page 1539
- **Process Monitoring** (option 168) with **SECTION MONITORING**

  Further information: "Process Monitoring (option 168)", Page 1236

Requirements

- Component Monitoring (software option 155)
- Components to be monitored are defined

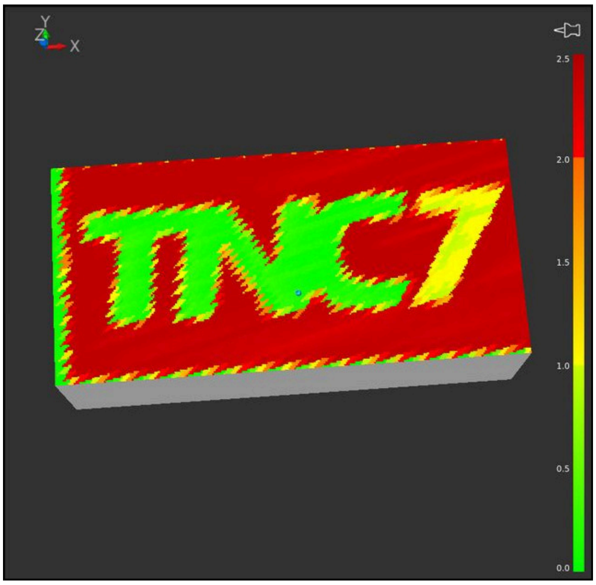
  In the optional machine parameter **CfgMonComponent** (no. 130900), the machine manufacturer defines the machine components to be monitored as well as the warning and error thresholds.

Description of function

A component heatmap is similar to the image from an infrared camera.

- Green: component works under conditions defined as safe
- Yellow: component works under warning zone conditions
- Red: Overload condition

The control shows these statuses on the workpiece in the simulation and can overwrite the statuses upon subsequent operations.



Representation of the component heat map in the simulation with missing pre-machining

Only one component at a time can be monitored with the heatmap. If you start the heatmap several times in a row, monitoring of the previous component is stopped.

Input

11 MONITORING HEATMAP START FOR "Spindle"	; Activate monitoring of the **Spindle** component and display it as a heat map

The NC function includes the following syntax elements:

Syntax element	Meaning
MONITORING HEATMAP	Syntax initiator for component monitoring
START FOR or **STOP**	Start or stop component monitoring
" " or **QS**	Fixed or variable name of the component to be monitored Only if **START FOR** is selected

Note

The control cannot display changes in the statuses directly in the simulation, as it must process the incoming signals (e.g. in the event of tool breakage). The control shows the change with a slight time delay.

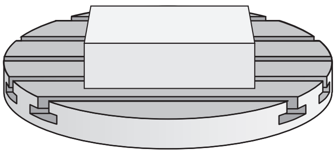
21.2 Cycles for monitoring

21.2.1 Cycle 239 ASCERTAIN THE LOAD (option 143)

ISO programming
G239

Application

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



The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of the table axes. With option 143 LAC (Load Adaptive Control) and Cycle **239 ASCERTAIN THE LOAD**, the control is able to automatically determine and adjust the actual mass inertia of the load, the actual friction forces, and the maximum axis acceleration or reset the feedforward and controller parameters. In this way, you can optimally react to major load changes. The control performs a weighing procedure to ascertain the weight acting on the axes. With this weighing run, the axes move by a specified distance. Your machine manufacturer defines the specific movements. Before weighing, the axes are moved to a position, if required, where there is no danger of collision during the weighing procedure. This safe position is defined by the machine manufacturer.

In addition to adjusting the control parameters, with LAC the maximum acceleration is also adjusted in accordance with the weight. This enables the dynamics to be accordingly increased with low load to increase productivity.

Cycle sequence

Parameter Q570 = 0

1 There is no physical movement of the axes.
2 The control resets the LAC.
3 The control activates feedforward and, if applicable, controller parameters that allow safe movements of the axis/axes, independently of the current load condition. The parameters set with **Q570**=0 are **independent** of the current load
4 These parameters can be useful during the setup procedure or after the completion of an NC program.

Parameter Q570 = 1

1 The control performs a weighing procedure in which it moves one or more axes. Which axes are moved depends on the setup of the machine and on the drives of the axes.
2 The scope of axis movement is defined by the machine manufacturer.
3 The feedforward and controller parameters determined by the control **depend** on the current load.
4 The control activates the ascertained parameters.

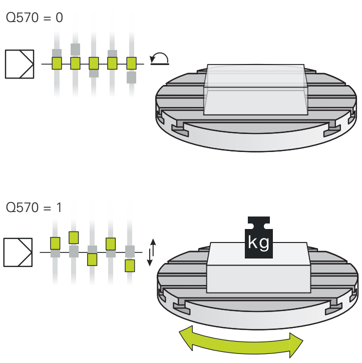
If you are using the mid-program startup function and the control thus skips Cycle **239** in the block scan, the control will ignore this cycle—no weighing run will be performed.

Notes

NOTICE
Danger of collision! This cycle may perform extensive movements in one or more axes at rapid traverse! There is a danger of collision! ▶ Contact your machine manufacturer to learn about the type and range of movements in Cycle **239** before using the cycle. ▶ Before the cycle starts, the control moves to a safe position, if applicable. The machine manufacturer determines this position. ▶ Set the potentiometers for feed-rate and rapid-traverse override to at least 50 % to ensure a correct ascertainment of the load.

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **239** becomes effective immediately after its definition.
- Cycle **239** supports the determination of the load on synchronized axes (gantry axes) if they have only one common position encoder (torque master slave).

Cycle parameters

Help graphic	Parameter
	**Q570 Load (0 = Delete/1 = Ascertain)?** Define whether the control will perform a LAC (Load Adaptive Control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters will be reset: **0**: Reset LAC; the values most recently ascertained by the control are reset, and the control uses load-independent feedforward and controller parameters **1**: Perform a weighing run; the control moves the axes and thus ascertains the feedforward and controller parameters depending on the current load. The values ascertained are activated immediately. Input: **0**, **1**

Example

```
11 CYCL DEF 239  ASCERTAIN THE LOAD ~
    Q570=+0                         ;LOAD ASCERTATION
```

21.2.2 Cycle 238 MEASURE MACHINE STATUS (option 155)

ISO programming
G238

Application

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

During their lifecycle, the machine components which are subject to loads (e.g., guides, ball screws, ...) become worn and thus, the quality of the axis movements deteriorates. This, in turn, affects the production quality.

Using **Component Monitoring** (Option 155) and Cycle **238**, the control is able to measure the current machine status. As a result, any deviations from the machine's shipping condition due to wear and aging can be measured. The measurement results are stored in a text file that is readable for the machine manufacturer. He can read and evaluate the data, and react with predictive maintenance, thereby avoiding unplanned machine downtimes.

The machine manufacturer can define warning and error thresholds for the measured values and optionally specify error reactions.

Related topics

- Component monitoring with **MONITORING HEATMAP** (option 155)

  Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1230

Cycle sequence

Ensure that the axes are not clamped before you start the measurement.

Parameter Q570=0

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are effective

Your machine manufacturer defines in detail how the axes will move.

Parameter Q570=1

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are **not** effective
3 On the **MON** status tab, you can select the monitoring task to be displayed
4 This diagram allows you to watch how close the components are to a warning or error threshold

Further information: "MON tab (option 155)", Page 177

Your machine manufacturer defines in detail how the axes will move.

Notes

NOTICE

Danger of collision!

This cycle may perform extensive movements in one or more axes at rapid traverse! If you program the cycle parameter **Q570** = 1, the feed rate and rapid traverse potentiometers, and, if applicable, the spindle potentiometer, have no effect. However, you can stop any movement by setting the feed rate potentiometer to zero. There is a danger of collision!

- Before recording measured data, test the cycle in test mode with **Q570 = 0**
- Contact your machine manufacturer to learn about the type and range of movements in Cycle **238** before using the cycle.

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **238** is CALL-active.
- If, during a measurement, you set, for example, the feed rate potentiometer to zero, then the control will abort the cycle and display a warning. You can acknowledge the warning by pressing the **CE** key and then press the **NC start** key to run the cycle again.

Cycle parameters

Help graphic	Parameter
	Q570 Mode (0=test/1=measure)? Define whether the control will perform a measurement of the machine status in test mode or in measurement mode: **0**: No measured data will be generated. You can control the axis movements with the feed rate and rapid traverse potentiometers **1**: This mode will generate measured data. You **cannot** control the axis movements with the feed rate and rapid traverse potentiometers Input: **0**, **1**

Example

```
11 CYCL DEF 238  MEASURE MACHINE STATUS ~
    Q570=+0                         ;MODE
```

21.3 Process Monitoring (option 168)

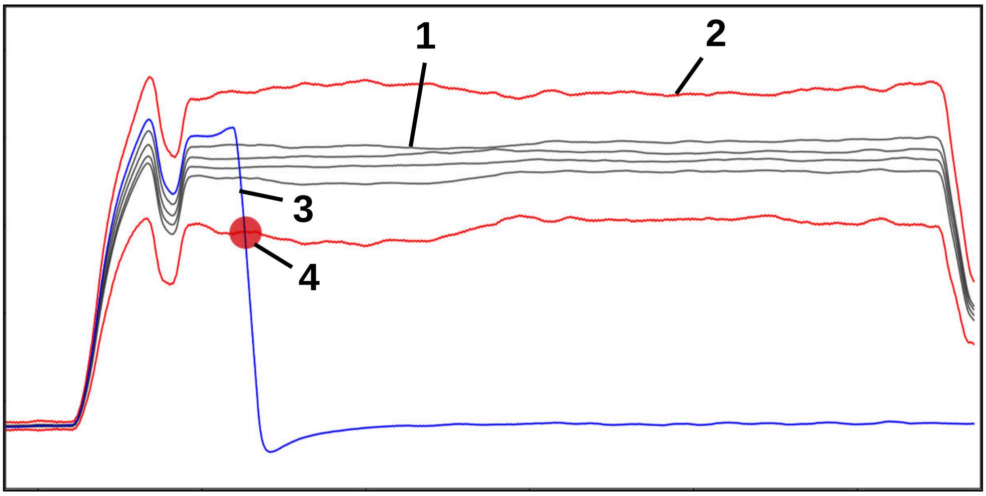
21.3.1 Fundamentals

The control uses process monitoring to detect disturbances in the machining process, e.g:.

- Tool breakage
- Incorrect or missing workpiece pre-machining
- Changed position or size of the workpiece blank
- Wrong material, e.g. aluminum instead of steel

Process monitoring allows you to monitor the machining process during program run using monitoring tasks. The monitoring task compares the signal curve of the current execution of an NC program with one or more reference machining operations. The monitoring task uses these reference machining operations to determine an upper and lower limit. If the current machining operation is outside the limits for a predefined hold time, the monitoring task executes a defined response. If, for example, the spindle current drops due to a tool breakage, the monitoring task executes a predefined response.

Further information: "Interrupting, stopping or canceling program run", Page 1959



Drop in spindle current due to tool breakage

1	References
2	Limits consisting of tunnel width and, if necessary, expansion
3	Current machining operation
4	A process fault (e.g., due to tool breakage)

If you are using process monitoring, the following steps are required:

- Defining monitoring sections in the NC program

  Further information: "Defining monitoring sections with MONITORING SECTION (option 168)", Page 1261
- Slowly running-in the NC program in Single Block mode before activating process monitoring

  Further information: "Program Run", Page 1953
- Activating process monitoring

  Further information: "Monitoring options column", Page 1254
- Running the NC program in Full Sequence

  Further information: "Program Run", Page 1953
- If necessary, configuring settings for the monitoring tasks
  - Selecting a strategy template

    Further information: "Strategy template", Page 1244
  - Adding or removing monitoring tasks

    Further information: "Icons", Page 1239
  - Defining settings and responses within the monitoring tasks

    Further information: "Monitoring task settings", Page 1246
  - Displaying monitoring task in the simulation as a process heat map

    Further information: "Monitoring options column within a monitoring section", Page 1255

    Further information: "Workpiece options column", Page 1539
- Running the NC program again in Full Sequence operating mode

  Further information: "Program Run", Page 1953
- Selecting other references and optimizing parameters

  Further information: "Monitoring tasks", Page 1245

  Further information: "Records of monitoring sections", Page 1257

Related topics

- **Component monitoring** (option 155) with **MONITORING HEATMAP**

  Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1230

21.3.2 Process Monitoring workspace (option 168)

Application

In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.

Further information: "Monitoring tasks", Page 1245

Requirements

- Process Monitoring (software option 168)
- Monitoring sections have been defined with **MONITORING SECTION**

  Further information: "Defining monitoring sections with MONITORING SECTION (option 168)", Page 1261
- Reproducibility of the process in the **FUNCTION MODE MILL** machining mode

  The monitoring tasks **FeedOverride** and **SpindleOverride** are functional in the **FUNCTION MODE TURN** machining mode (option 50).

Functionality

The **Process Monitoring** workspace provides information and settings for monitoring the machining process.

Depending on the cursor position in the NC program, the control provides the following areas:

- Global area

  The control shows information about the active NC program.

  Further information: "Global area", Page 1241
- Strategy area

  The control shows the monitoring tasks and the graphs of the recordings. You can configure settings for the monitoring tasks.

  Further information: "Strategy area", Page 1243
- **Monitoring options** column in the global area

  The control displays information on the recordings that relate to all monitoring sections of the NC program.

  Further information: "Monitoring options column in the global area", Page 1255
- **Monitoring options** column within a monitoring section

  The control displays information on the recordings that relate only to the currently selected monitoring section.

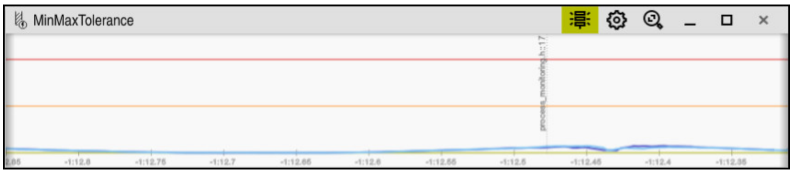
  Further information: "Monitoring options column within a monitoring section", Page 1255

Icons

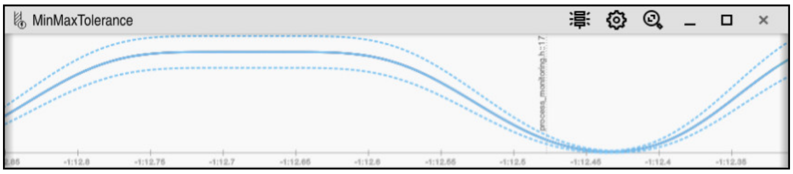
The following icons are shown in the **Process Monitoring** workspace:

Icon	Meaning
	Show or hide the **Monitoring options** column **Further information:** "Monitoring options column", Page 1254
	Switch on/off setup mode If setup mode is active, the control displays the settings for process monitoring. For running the program, you can switch off setup mode.
	Remove monitoring task **Further information:** "Monitoring tasks", Page 1245 Available only in setup mode
	Add monitoring task **Further information:** "Monitoring tasks", Page 1245 Available only in setup mode
	Open settings You can open the following settings: ■ **Process Monitoring** workspace setting **Further information:** "Settings for the Process Monitoring workspace", Page 1253 ■ Setting in the **Settings for NC program** window of the **Monitoring options** column **Further information:** "Settings for NC program window", Page 1260 Available only in setup mode ■ Monitoring task setting **Further information:** "Monitoring task settings", Page 1246 Available only in setup mode
	Set the graph size to 100%

Icon	Meaning
	Show or hide warning and error limits If you show the warning and error limits, the control shows the monitored signal in relation to the defined limits. The control shows the following warning and error limits: ■ Green line If the current machining operation is at the bottom line, the current machining operation corresponds to the reference. ■ Orange line This line shows the warning limit. If the current machining operation exceeds the middle line, the current machining operation deviates by half the set limit of the reference. ■ Red line This line shows the error limit. If the current machining operation exceeds the upper line for a defined hold time, the monitoring task triggers a defined response (e.g., NC stop). If you hide the warning and error limits, the control shows an absolute display of the monitored signal. The dashed lines represent the upper and lower error limits, i.e. the tunnel width.



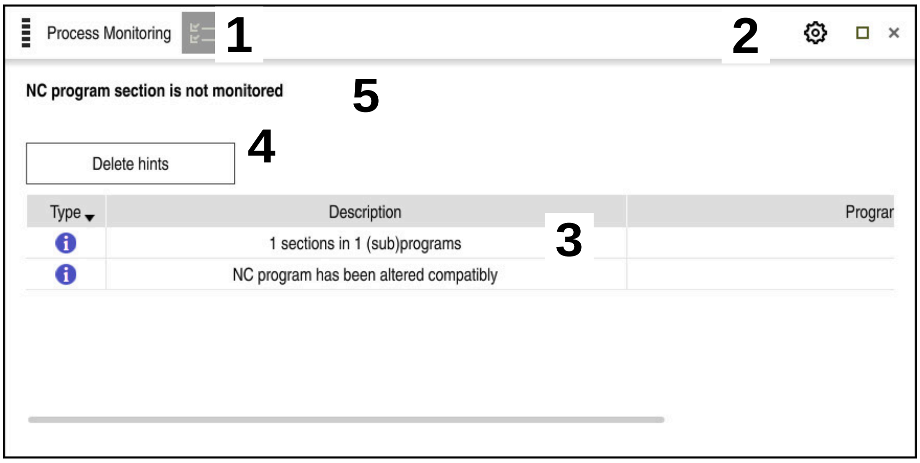
Warning and error limits displayed: The control shows the signal in relation to the defined limits



Warning and error limits hidden: The solid line represents the signal and the dashed lines represent the tunnel width determined at the time

Global area

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the global area.



Global area in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the global area:

1 **Monitoring options** icon
   Further information: "Monitoring options column", Page 1254
2 **Settings** icon for the **Process Monitoring** workspace
   Further information: "Settings for the Process Monitoring workspace", Page 1253
3 Table with notes on the active NC program
   Further information: "Notes on the NC program", Page 1242
4 **Delete hints** button
   You can use the **Delete hints** button to empty the table.
5 Information that this area is not monitored in the NC program

Notes on the NC program

In this area, the control shows a table with information about the active NC program. The table contains the following information:

Column or symbol	Meaning
Type	In the **Type** column, the control shows different types of notifications.
	Information (for example, the number of monitoring sections)
	Warning (for example, whether a monitoring section has been removed)
	Error (for example, whether you should reset the recordings) If you make changes within a monitoring section, that monitoring section can no longer be monitored. Therefore, you should reset the recordings and set new references so that machining is monitored again. **Further information:** "Settings for NC program window", Page 1260 You can sort the table by information type by selecting the **Type** column.
Description	In the **Description** column, the control displays information about the information types, e.g.: ■ Changes to the NC program ■ Cycles contained in the NC program ■ Interruptions (e.g., **M0** or **M1**)
Program line	If the information depends on an NC block number, the control displays the program name and the NC block number.

Strategy area

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the strategy area.

Strategy area in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the strategy area:

1 **Monitoring options** icon
   Further information: "Monitoring options column", Page 1254
2 Switch on/off setup mode
   Further information: "Icons", Page 1239
3 **Settings** icon for the **Process Monitoring** workspace
   Further information: "Settings for the Process Monitoring workspace", Page 1253
4 **Settings** icon for the monitoring tasks
   Further information: "Monitoring task settings", Page 1246
   Available only in setup mode
5 Show or hide warning and error limits
   Further information: "Icons", Page 1239
6 Monitoring tasks
   Further information: "Monitoring tasks", Page 1245

7 The control shows the following information and functions:
- Name of the monitoring section, if applicable

  If **AS** is defined in the NC program with the optional syntax element, the control displays the name.

  If no name is defined, the control displays **MONITORING SECTION**.

  Further information: "Input", Page 1262
- Range of NC block numbers of the monitoring section in square brackets

  Start and end of the monitoring section in the NC program
- **Original strategy** or **Save strategy as template** button

  Further information: "Strategy template", Page 1244
- Selection menu for strategy template

  Further information: "Strategy template", Page 1244

Available only in setup mode

Strategy template

A strategy template includes one or more monitoring tasks, including the defined settings.

You can choose between the following strategy templates using a selection menu:

Strategy template	Meaning
MinMaxTolerance	This strategy template includes the following monitoring tasks: ■ **MinMaxTolerance** **Further information:** "Monitoring task MinMaxTolerance", Page 1247 ■ **SignalDisplay** **Further information:** "Monitoring task SignalDisplay", Page 1251 ■ **SpindleOverride** **Further information:** "Monitoring task SpindleOverride", Page 1251 ■ **FeedOverride** **Further information:** "Monitoring task FeedOverride", Page 1252
StandardDeviation	This strategy template includes the following monitoring tasks: ■ **StandardDeviation** **Further information:** "Monitoring task StandardDeviation", Page 1250 ■ **SignalDisplay** **Further information:** "Monitoring task SignalDisplay", Page 1251 ■ **SpindleOverride** **Further information:** "Monitoring task SpindleOverride", Page 1251 ■ **FeedOverride** **Further information:** "Monitoring task FeedOverride", Page 1252

Strategy template	Meaning
User-defined	In this strategy template, you can compile the monitoring tasks yourself.

If you modify a strategy template, you can overwrite the modified strategy template by clicking the **Save strategy as template** button. The control overwrites the currently selected strategy template.

> Since you cannot restore the as-delivered state of the strategy templates yourself, only overwrite the **User-defined** template.
>
> The machine manufacturer can use the optional machine parameter **ProcessMonitoring** (no. 133700) to restore the as-delivered state of the strategy templates.

In the settings of the **Process Monitoring** workspace, you can define which strategy template the control selects by default after creating a new monitoring section.

Further information: "Settings for the Process Monitoring workspace", Page 1253

Monitoring tasks

The **Process Monitoring** workspace contains the following monitoring tasks:

- **MinMaxTolerance**

  With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references, including the predefined percentage and static deviation.

  Further information: "Monitoring task MinMaxTolerance", Page 1247
- **StandardDeviation**

  With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including static expansion and a multiple of the standard deviation σ.

  Further information: "Monitoring task StandardDeviation", Page 1250
- **SignalDisplay**

  With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

  Further information: "Monitoring task SignalDisplay", Page 1251
- **SpindleOverride**

  With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

  Further information: "Monitoring task SpindleOverride", Page 1251
- **FeedOverride**

  With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

  Further information: "Monitoring task FeedOverride", Page 1252

In each monitoring task, the control shows the current processing and the selected references as a graph. The time axis is specified in seconds, or in minutes for longer monitoring sections.

Monitoring task settings

You can change the settings of the monitoring tasks for each monitoring section. When you select the settings of a monitoring task, the control displays two areas. In the area on the left, the settings that were active at the time of the selected recording are grayed out. In the area on the right, the current monitoring task settings are shown. The **Apply** button allows you to save the settings from the area on the left or right. You can also remove a monitoring task from a monitoring section or add one using the plus sign.

The default values of the monitoring tasks are recommended initial values. These initial values can be adjusted for your machining purposes.

If you change the settings of a monitoring task or add a new monitoring task, the changes are identified by the ***** character preceding the name.

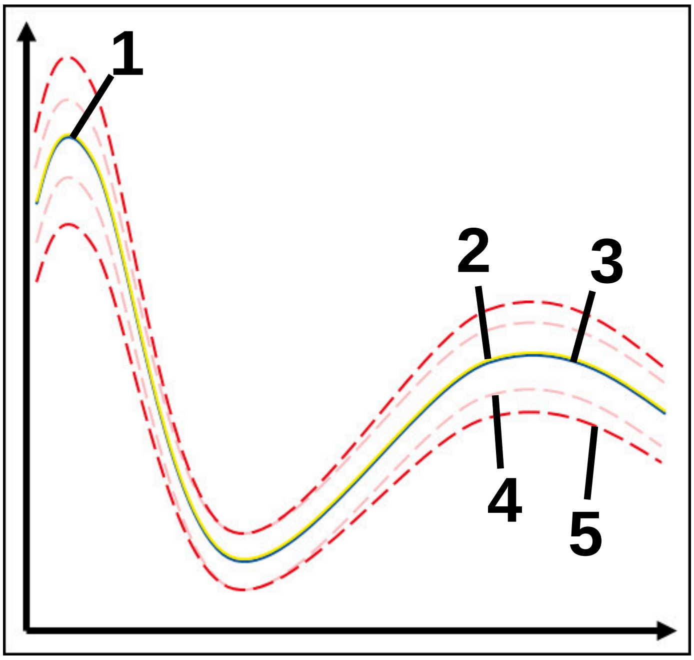
Monitoring task MinMaxTolerance

With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references, including the predefined percentage and static deviation.

The application cases of **MinMaxTolerance** are significant process faults (e.g., during small series production):

- Tool breakage
- Missing tool
- Changed position or size of the workpiece blank

The control requires at least one recorded machining operation as a reference. If you do not select a reference, this monitoring task is inactive and does not draw a graph.



1		First good reference
2		Second good reference
3		Third good reference
4		Limits consisting of the tunnel width
5		Limits consisting of a percentage expansion of the static tunnel width

Further information: "Records of monitoring sections", Page 1257

If, for example, you have a recording that is only just acceptable due to tool wear, you can also use an alternative application with this monitoring task.

Further information: "Alternative application with acceptable reference", Page 1249

Settings for MinMaxTolerance

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**
  Percentage expansion of tunnel width
- **Static tunnel width**
  Upper and lower limits, based on references
- **Hold time**
  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Trigger warning**
  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.
  Further information: "Message menu on the information bar", Page 1532
- **Trigger NC stop**
  If the signal exceeds the limits for more than the defined hold time, the control stops the NC program. You can then check the machining status. If you find that there is no serious error, you can resume the NC program.
- **Abort program run**
  If the signal exceeds the limits for more than the defined hold time, the control aborts the NC program. In this case, the NC program cannot be resumed.
- **Lock the tool**
  If the signal exceeds the warning limits for more than the defined hold time, the control locks the tool in tool management.
  Further information: "Tool management ", Page 297

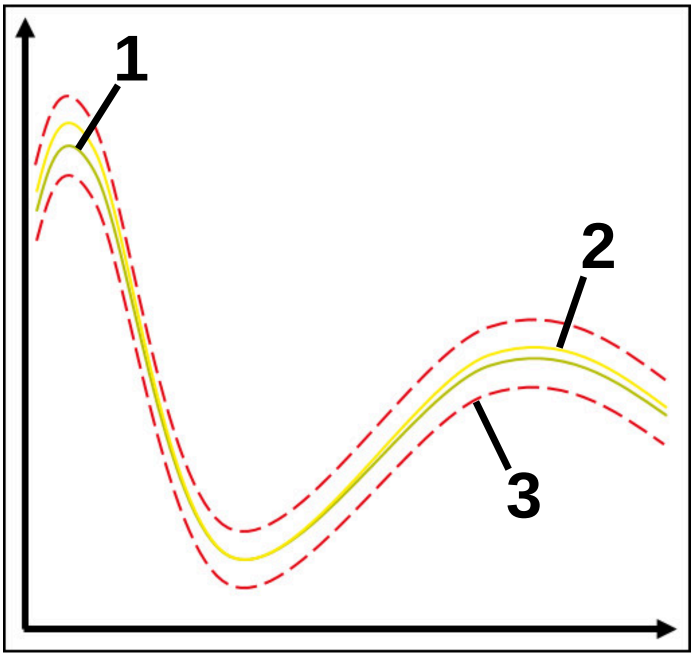
Alternative application with acceptable reference

If the control has recorded a machining operation that is only just acceptable, you can use an alternative application of the monitoring task **MinMaxTolerance**.

Select at least two references:

- An optimal reference
- A reference that is only just acceptable, e.g. showing a higher signal of the spindle load due to tool wear

The monitoring task checks whether the current machining operation is within the range of the selected references. For this strategy, select no deviation or a low percentage deviation, since the tolerance is already given by the different references.



1	Optimal reference
2	Reference only just acceptable
3	Limits consisting of the tunnel width

Monitoring task StandardDeviation

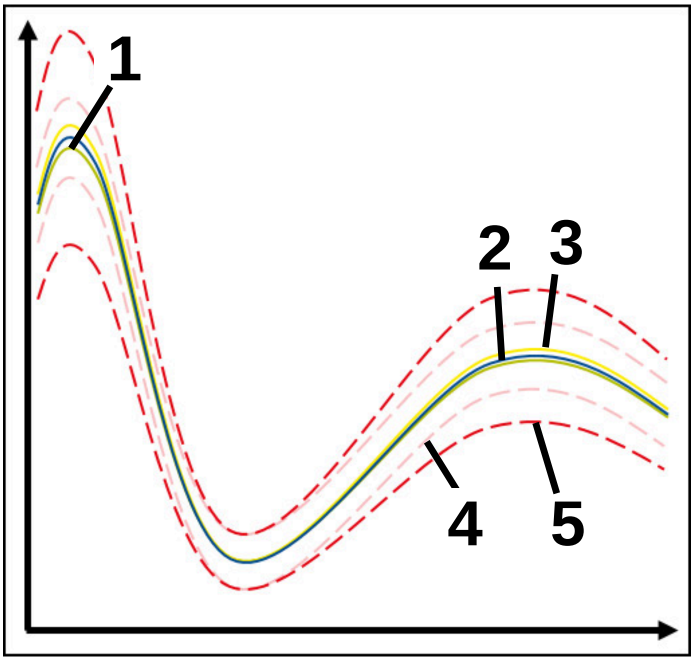
With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including static expansion and a multiple of the standard deviation σ.

The application cases of **StandardDeviation** are process faults of all kinds (e.g., during series production):

- Tool breakage
- Missing tool
- Tool wear
- Changed position or size of the workpiece blank

The control requires at least three recorded machining operations as references. The references should include one optimal, one good and one only just acceptable machining operation. If you do not select the required references, this monitoring task is not active and does not draw a graph.

Further information: "Records of monitoring sections", Page 1257



1		Optimal reference
2		Good reference
3		Reference only just acceptable
4		Limits consisting of the tunnel width
5		Limits consisting of the expansion of the tunnel width multiplied by factor σ

Settings for StandardDeviation

You can use sliders to configure the following settings for this monitoring task:

- **A multiple of σ**

  Expansion of the tunnel width multiplied by factor σ
- **Static tunnel width**

  Upper and lower limits, based on references
- **Hold time**

  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Trigger warning**

  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.

  Further information: "Message menu on the information bar", Page 1532
- **Trigger NC stop**

  If the signal exceeds the limits for more than the defined hold time, the control stops the NC program. You can then check the machining status. If you find that there is no serious error, you can resume the NC program.
- **Abort program run**

  If the signal exceeds the limits for more than the defined hold time, the control aborts the NC program. In this case, the NC program cannot be resumed.
- **Lock the tool**

  If the signal exceeds the warning limits for more than the defined hold time, the control locks the tool in tool management.

  Further information: "Tool management ", Page 297

Monitoring task SignalDisplay

With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

You can compare whether the current machining operation corresponds to the references. This allows you to visually check whether you can use the machining operation as a reference.

The monitoring task does not respond.

Monitoring task SpindleOverride

With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

The control uses the first recorded machining operation as a reference.

Settings for SpindleOverride

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**
  Accepted deviation of the override in percent compared to the first recording
- **Hold time**
  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Trigger warning**
  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.
  Further information: "Message menu on the information bar", Page 1532
- **Trigger NC stop**
  If the signal exceeds the limits for more than the defined hold time, the control stops the NC program. You can then check the machining status. If you find that there is no serious error, you can resume the NC program.

Monitoring task FeedOverride

With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

The control uses the first recorded machining operation as a reference.

FeedOverride settings

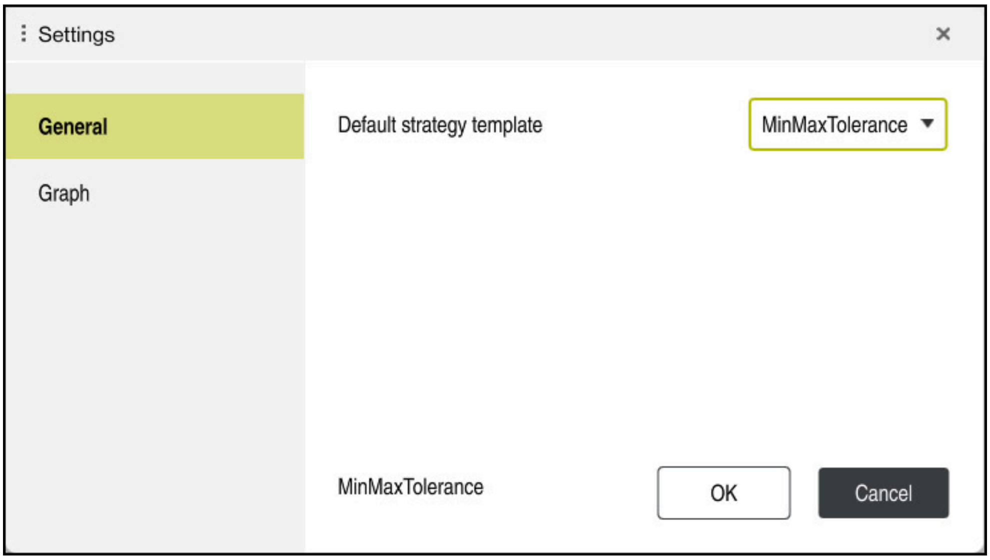
You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**
  Accepted deviation of the override in percent compared to the first recording
- **Hold time**
  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Trigger warning**
  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.
  Further information: "Message menu on the information bar", Page 1532
- **Trigger NC stop**
  If the signal exceeds the limits for more than the defined hold time, the control stops the NC program. You can then check the machining status. If you find that there is no serious error, you can resume the NC program.

Settings for the Process Monitoring workspace



Settings for the **Process Monitoring** workspace

General

In the **General** area, select which strategy template the control uses as the default:

- **MinMaxTolerance**
- **StandardDeviation**
- **User-defined**

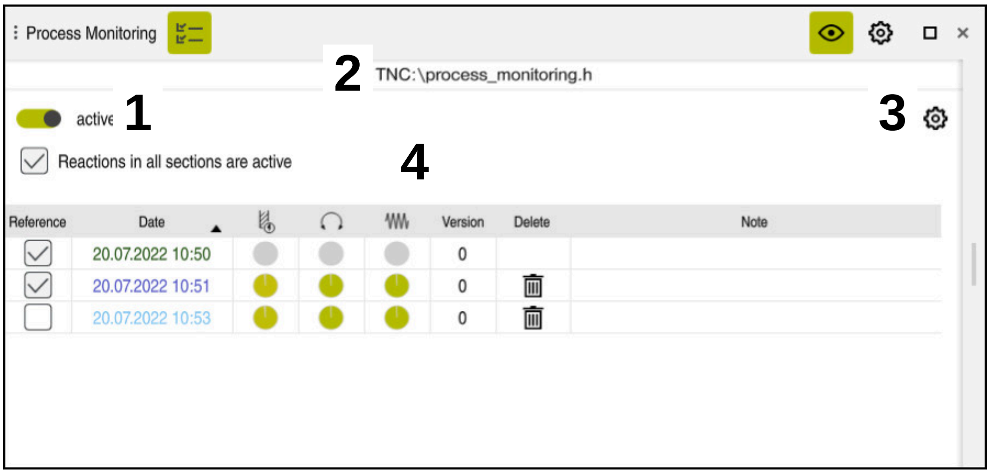
Further information: "Strategy template", Page 1244

Graph

In the **Graph** area, you can select the following settings:

Setting	Meaning
Simultaneously plotted references	Select the maximum number of recordings that the control displays simultaneously as graphs in the monitoring tasks: ■ **2** ■ **4** ■ **6** ■ **8** ■ **10** If more references are selected than the control is to display, the last selected references will be displayed as recordings.
Preview [s]	The control can run selected references as a preview during program execution. The control then shifts the time axis of the machining operation to the left. Select how many seconds of the reference the control will preview: ■ **0** ■ **2** ■ **4** ■ **6** **Further information:** "Records of monitoring sections", Page 1257

Monitoring options column



Monitoring options column in the global range

The **Monitoring options** column shows the following in the upper area regardless of the cursor position in the NC program:

1 Toggle switch for activating or deactivating process monitoring for the entire NC program
2 Path of the current NC program
3 Open the **Settings** icon in the **Settings for NC program** window
   Further information: "Settings for NC program window", Page 1260
   Available only in setup mode
4 Checkbox for activating or deactivating the responses of all monitoring sections in the NC program
   Available only in setup mode

Depending on the cursor position in the NC program, the control provides the following areas:

- **Monitoring options** column in the global area
  You can select references that are effective for all monitoring sections of the NC program.
  Further information: "Monitoring options column in the global area", Page 1255
- **Monitoring options** column within a monitoring section
  You can define settings and select references that apply to the currently selected monitoring section.
  Further information: "Monitoring options column within a monitoring section", Page 1255

Monitoring options column in the global area

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column in the global area.

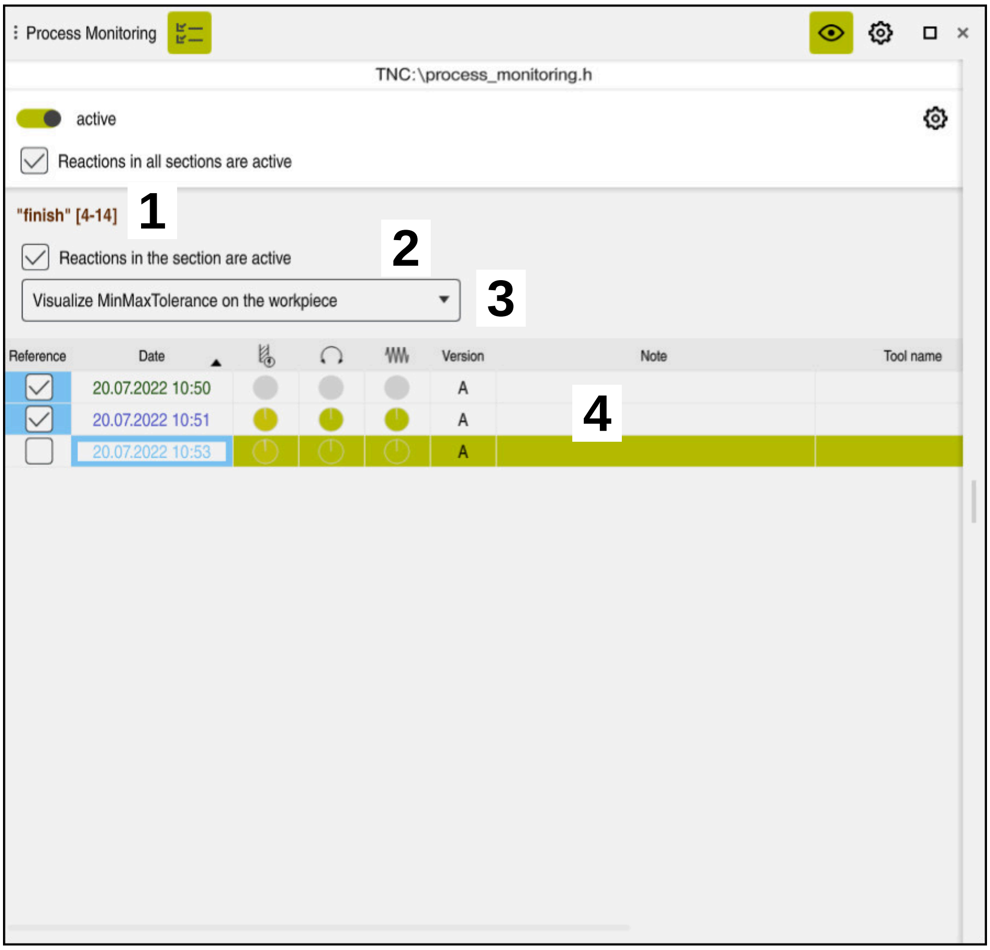
In the global area, the control displays a table with the records of all monitoring sections of the NC program.

Further information: "Records of monitoring sections", Page 1257

Monitoring options column within a monitoring section

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column within the monitoring section.

If the cursor is within the monitoring section, the control will leave this area gray.



Monitoring options column within the monitoring section

The **Monitoring options** column displays the following within the monitoring section:

1. The control shows the following information and functions:
   - Name of the monitoring section, if applicable

     If **AS** is defined in the NC program with the optional syntax element, the control displays the name.

     If no name is defined, the control displays **MONITORING SECTION**.

     Further information: "Input", Page 1262
   - Range of NC block numbers of the monitoring section in square brackets

     Start and end of the monitoring section in the NC program
2. Checkbox for activating and deactivating the responses in the monitoring section

   You can activate or deactivate the responses of the currently selected monitoring section.

   Available only in setup mode
3. Selection menu for the process heat map

   You can display a monitoring task as a process heat map in the **Simulation** workspace.

   Further information: "Workpiece options column", Page 1539

   Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1230

   Available only in setup mode
4. Table with the recordings of the monitoring section

   The recordings refer only to the monitoring section in which the cursor is currently located.

   Further information: "Records of monitoring sections", Page 1257

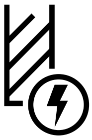
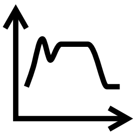
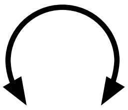
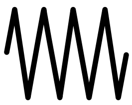
Records of monitoring sections

The contents and functions of the table with the recordings of the machining operations depend on the cursor position in the NC program.

Further information: "Monitoring options column", Page 1254

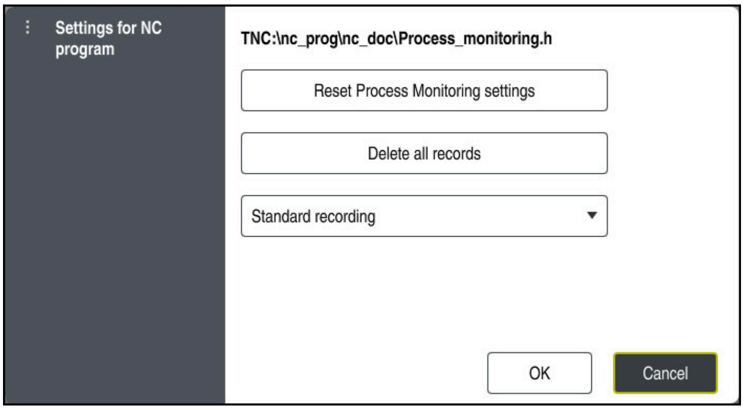
The table contains the following information about the monitoring section:

Column	Information or action
Reference	If you activate the checkbox for a table row, the control uses this recording as a reference for the corresponding monitoring tasks. If you activate multiple table rows, the control uses all selected rows as references. If you select multiple references with greater deviation, the tunnel width also increases. You can select up to ten references at a time. The effect of the reference depends on the position of the cursor in the NC program: ■ Within the monitoring section: The reference only applies to the currently selected monitoring section. The control shows a dash in the global range in this table row for information. If a table row is marked as a reference in all strategy areas or in the global area, the control displays a check mark. ■ Global area: The reference applies to all monitoring sections of the NC program. Mark recordings that have a provided satisfactory result, such as a clean surface, as a reference. You can only select a recording of a complete machining operation as a reference. When you select a recording, the control color-highlights the references selected for this recording in this column.
Date	The control displays the date and time of the program start or the starting time of the monitoring section of each recorded machining operation. If you select the **Date** column, the control sorts the table by date.

Column	Information or action
   	The control displays a color representation of the coverage of each monitoring task. The coverage indicates the percentage to which the graph of the recording corresponds to the reference graph. Warning and error limits are displayed in color. When you select a row in this column, the control displays the coverage as a percentage. If setup mode is active, the control displays the coverage as a pie chart. If the coverage is at about 80%, the machining operation is still OK. In case of lower values, make sure to check the machining process. Coverage depends on the following factors: ■ Time delay (e.g., change of the feed-rate override) If the potentiometer position of the feed-rate override deviates from the reference machining operation, the coverage becomes worse. ■ Local delay (e.g., due to tool compensation with **DR**) If the path of the tool center point (**TCP**) deviates from the reference machining operation, the coverage becomes worse. **Further information:** "Tool center point (TCP, tool center point)", Page 273 In this column, the control displays notes on the responses defined for the monitoring tasks. When you select a table cell that contains a note, the control displays detailed information on the response.
Version	If you defined custom process-monitoring settings, the control displays a different version in this column. In the **Version** column, the control displays the following information, depending on the area: ■ Within the monitoring section: The control displays letters for the different versions within the monitoring section. ■ Global area: The control displays numbers for the different versions within at least one monitoring section. Available only in setup mode
Delete	If you select the trash bin icon, the control deletes the table row and the associated recorded process data. You cannot delete the first row of the table because this row is used as a reference for the following functions: ■ For the Quality column ■ **SpindleOverride** monitoring task ■ **FeedOverride** monitoring task You delete all recordings including the first in the **Settings for NC program** window. Only in the global area
Note	In the **Note** column, you can enter notes about the table row.
Tool name	Name of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 297

Column	Information or action
R	Radius of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 297
DR	Delta value of the tool radius from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 297
L	Length of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 297
CUT	Number of cutting edges of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 297
CURR_TIME	Tool life from the tool management at the start of the respective machining operation Only within the monitoring section **Further information:** "Tool management ", Page 297

Settings for NC program window



Settings for NC program window

The **Settings for NC program** window provides the following settings:

- **Reset Process Monitoring settings**
- **Delete all records**, including the first table row
- Selection menu that displays the type and number of recorded machining operations
  - **Standard recording**

    The control records all information.
  - **Limit recordings**

    The control records all machining operations up to a certain count.

    If the number of machining operations exceeds the maximum number, the control overwrites the last machining operation.

    Input: **2...999999999**
  - **Only meta-information**

    The control does not record any process data, but only meta-information, such as the date and time. This means that you cannot use this recording as a reference. This setting can be used for monitoring and logging once process monitoring has been set up completely. This setting significantly reduces the amount of data.
  - **Each nth recording**

    The control does not record process data for each machining operation. You can define after which number of machining operations the control records process data. For the other machining operations, only meta-information will be recorded.

    Input: **2...20**

  Further information: "Records of monitoring sections", Page 1257

Notes

- If you use different sizes of workpiece blanks, set process monitoring to a more tolerant setting or start the first monitoring section after pre-machining.
- If the spindle load is too low, the control may not detect any difference from idling (e.g., for a tool with a small diameter).
- If you remove and add a monitoring task again, the previous recordings remain.
- The machine manufacturer can define the behavior of the control in connection with pallet machining in case a program is aborted (e.g., continue machining the workpieces on the next pallet).

Basics of operation

- You can zoom in or out of the graph horizontally by dragging or scrolling.
- If you drag or swipe with the left mouse button held down, you can move the graph.
- You can align the graph by selecting an NC block number. The control marks the selected NC block number in green within the monitoring task.
- If you double-tap or double-click a position within the graph, the control selects the corresponding NC block in the program.

  Further information: "Common gestures for the touchscreen", Page 116

21.3.3 Defining monitoring sections with MONITORING SECTION (option 168)

Application

With the **MONITORING SECTION** function, you divide the NC program into monitoring sections for process monitoring.

Related topics

- **Process Monitoring** workspace

  Further information: "Process Monitoring workspace (option 168)", Page 1238

Requirement

- Process Monitoring (software option 168)

Description of function

MONITORING SECTION START is used to define the start of a new monitoring section and **MONITORING SECTION STOP**, to define the end of the monitoring section.

You must not nest monitoring sections.

If you do not define a **MONITORING SECTION STOP**, the control still interprets a new monitoring section for the following functions:

- For a new **MONITORING SECTION START**
- For a physical **TOOL CALL**

  The control only interprets a new monitoring section for a tool call when a tool change takes place.

  Further information: "Tool call by TOOL CALL", Page 304

If you program the following syntax elements, the control displays a note:

- Positions relative to the machine datum (e.g., **M91**)
- Call of a replacement tool with **M101**
- Automatic liftoff with **M140**
- Repeats with variable values (e.g., **CALL LBL 99 REP QR1**)
- Jump commands (e.g., **FN 5**)
- Spindle-related M functions (e.g., **M3**)
- New monitoring section defined by **TOOL CALL**
- Monitoring section ended by **PGM END**

Further information: "Notes on the NC program", Page 1242

If you program the following syntax elements, the control displays an error:

- Syntax error within a monitoring section
- Stop within the monitoring section (e.g., **M0**)
- Call of an NC program within the monitoring section (e.g., **PGM CALL**)
- Missing subprograms
- End of a monitoring section precedes the start of that monitoring section
- Multiple monitoring sections with the same contents

If an error is displayed, process monitoring cannot be used.

Further information: "Notes on the NC program", Page 1242

Input

11 MONITORING SECTION START AS "finish contour"	; Start of the monitoring section, specifying an additional name

The NC function includes the following syntax elements:

Syntax element	Meaning
MONITORING SECTION	Syntax initiator for the monitoring section of process monitoring
START or **STOP**	Start or end of the monitoring section
AS	Additional designation Optional syntax element Only when **START** is selected

Notes

- The control shows the start and end of the monitoring section in the structure.
  Further information: "Settings in the Program workspace", Page 220
- End the monitoring section before the end of the program with **MONITORING SECTION STOP**.
  If you do not define an end for the monitoring section, the control ends the monitoring section with **END PGM**.
- Monitoring sections used for process monitoring must not overlap with the **AFC** sections.
  Further information: "Adaptive Feed Control (AFC, option 45)", Page 1196

22

Multiple-Axis Machining

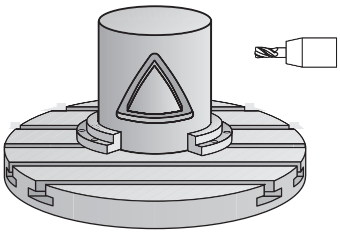
22.1 Cycles for cylinder surface machining

22.1.1 Cycle 27 CYLINDER SURFACE (option 8)

ISO programming
G127

Application

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



This cycle enables you to program a contour in two dimensions and then transfer it onto a cylindrical surface. Use Cycle **28** to mill guide slots on the cylinder.

Describe the contour in a subprogram that you program with Cycle **14 CONTOUR**.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

The coordinate data of the unrolled cylinder surface (X coordinates), which define the position of the rotary table, can be entered as desired either in degrees or in mm (or inches) (**Q17**).

Cycle sequence

1 The control positions the tool above the cutter infeed point, taking the finishing allowance for side into account
2 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate **Q12**.
3 At the end of the contour, the control returns the tool to set-up clearance and returns to the infeed point
4 Steps 1 to 3 are repeated until the programmed milling depth **Q1** is reached.
5 Subsequently, the tool retracts in the tool axis to the clearance height.

	The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

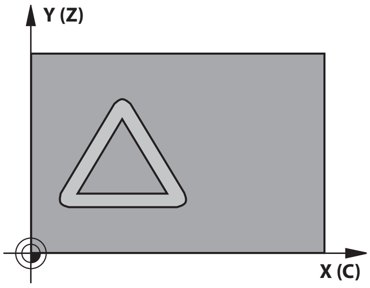
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.
- This cycle can also be used in a tilted working plane.

> The machining time can increase if the contour consists of many non-tangential contour elements.

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	**Q1 Milling depth?** Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**

Example

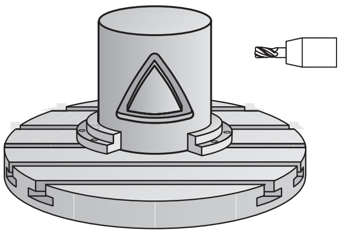
11 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-20	;MILLING DEPTH ~
Q3=+0	;ALLOWANCE FOR SIDE ~
Q6=+0	;SET-UP CLEARANCE ~
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q16=+0	;RADIUS ~
Q17=+0	;TYPE OF DIMENSION

22.1.2 Cycle 28 CYLINDRICAL SURFACE SLOT (option 8)

ISO programming
G128

Application

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



With this cycle you can program a guide slot in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle **27**, with this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the slot are nearly parallel. You can machine exactly parallel walls by using a tool that is exactly as wide as the slot.

The smaller the tool is with respect to the slot width, the larger the distortion in circular arcs and oblique line segments. To minimize this process-related distortion, you can define the parameter **Q21**. This parameter specifies the tolerance with which the control machines a slot as similar as possible to a slot machined with a tool of the same width as the slot.

Program the center path of the contour together with the tool radius compensation. With the radius compensation you specify whether the control cuts the slot with climb milling or up-cut milling.

Cycle sequence

1 The control positions the tool above the infeed point.
2 The control moves the tool vertically to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate **Q12**. The approaching behavior depends on the **ConfigDatum CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004) parameter
3 At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate **Q12** while respecting the finishing allowance for the side
4 At the end of the contour, the control moves the tool to the opposite slot wall and returns to the infeed point.
5 Steps 2 to 3 are repeated until the programmed milling depth **Q1** is reached.
6 If you defined the tolerance in **Q21**, the control then re-machines the slot walls to be as parallel as possible
7 Finally, the tool retracts in the tool axis to the clearance height.

The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

NOTICE

Danger of collision!

At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle need not be the same as the starting position. There is a danger of collision!

- Control the traversing movements of the machine
- In the **Simulation** workspace of the **Editor** operating mode, check the end position of the tool after the cycle
- After the cycle, program the absolute (not incremental) coordinates

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.
- This cycle can also be used in a tilted working plane.

The machining time can increase if the contour consists of many non-tangential contour elements.

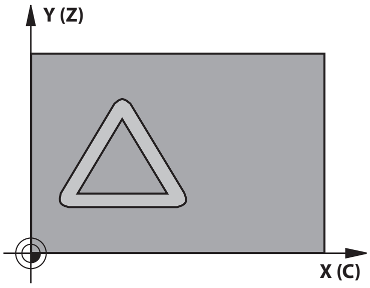
Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Note regarding machine parameters

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line

Cycle parameters

Help graphic	Parameter
	**Q1 Milling depth?** Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**
	Q20 Slot width? Width of the slot to be machined Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q21 Tolerance? If you use a tool smaller than the programmed slot width **Q20**, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance **Q21**, the control adds a subsequent milling operation to ensure that the slot dimensions are as close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With **Q21**, you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the re-machining takes. **Recommendation**: Use a tolerance of 0.02 mm. **Function inactive**: Enter 0 (default setting). Input: **0...9.9999**

Example

```
11 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~
    Q1=-20                  ;MILLING DEPTH ~
    Q3=+0                   ;ALLOWANCE FOR SIDE ~
    Q6=+2                   ;SET-UP CLEARANCE ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q16=+0                  ;RADIUS ~
    Q17=+0                  ;TYPE OF DIMENSION ~
    Q20=+0                  ;SLOT WIDTH ~
    Q21=+0                  ;TOLERANCE
```

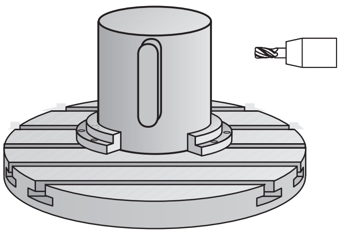
22.1.3 Cycle 29 CYL SURFACE RIDGE (option 8)

ISO programming

G129

Application

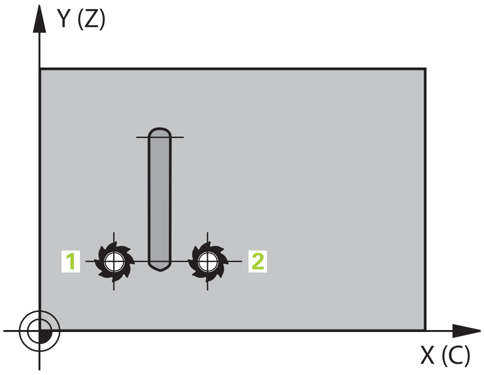
	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. With this cycle, the control adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the center path of the ridge together with the tool radius compensation. With the radius compensation you specify whether the control cuts the ridge with climb milling or up-cut milling.

At the ends of the ridge, the control will always add a semi-circle whose radius corresponds to half the ridge width.

Cycle sequence



1 The control positions the tool above the starting point of machining. The control calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins to the left (**1**, RL = climb milling) or to the right of the ridge (**2**, RR = up-cut milling).
2 After the control has positioned the tool to the first plunging depth, the tool moves on a circular arc at the milling feed rate **Q12** tangentially to the ridge wall. A finishing allowance programmed for the side is taken into account.
3 At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate **Q12** until the ridge is completed.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth **Q1** is reached.
6 Finally, the tool retracts in the tool axis to the clearance height.

The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**
	Q20 Ridge width? Width of the ridge to be machined Input: **-99999.9999...+99999.9999**

Example

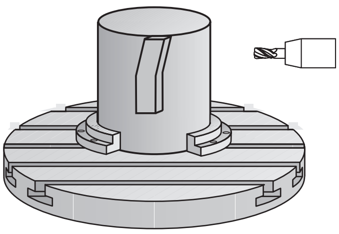
11 CYCL DEF 29 CYL SURFACE RIDGE ~
Q1=-20 ;MILLING DEPTH ~
Q3=+0 ;ALLOWANCE FOR SIDE ~
Q6=+2 ;SET-UP CLEARANCE ~
Q10=-5 ;PLUNGING DEPTH ~
Q11=+150 ;FEED RATE FOR PLNGNG ~
Q12=+500 ;FEED RATE F. ROUGHNG ~
Q16=+0 ;RADIUS ~
Q17=+0 ;TYPE OF DIMENSION ~
Q20=+0 ;RIDGE WIDTH

22.1.4 Cycle 39 CYL. SURFACE CONTOUR (option 8)

ISO programming
G139

Application

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



This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the unrolled surface of the cylinder. With this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the milled contour are always parallel to the cylinder axis.

Describe the contour in a subprogram that you program with Cycle **14 CONTOUR**.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

Unlike in Cycles **28** and **29**, in the contour subprogram, you define the contour actually to be machined.

Cycle sequence

1 The control positions the tool above the starting point of machining. The control locates the starting point next to the first point defined in the contour subprogram offset by the tool diameter
2 The control then moves the tool vertically to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate **Q12**. A finishing allowance programmed for the side is taken into account. The approach behavior depends on the machine parameter **apprDepCylWall** (no. 201004)
3 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate **Q12** until the contour train is complete.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth **Q1** is reached.
6 Finally, the tool retracts in the tool axis to the clearance height.

> The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

> This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

> - Ensure that the tool has enough space laterally for contour approach and departure.
> - The machining time can increase if the contour consists of many non-tangential contour elements.

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Note regarding machine parameters

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**

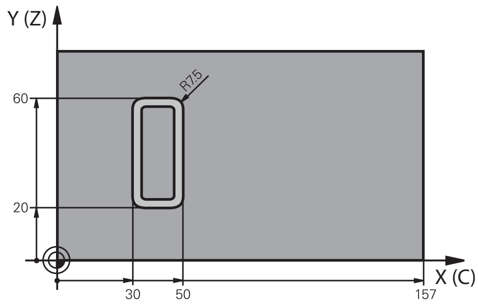
Example

```
11 CYCL DEF 39 CYL. SURFACE CONTOUR ~
    Q1=-20                  ;MILLING DEPTH ~
    Q3=+0                   ;ALLOWANCE FOR SIDE ~
    Q6=+2                   ;SET-UP CLEARANCE ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q16=+0                  ;RADIUS ~
    Q17=+0                  ;TYPE OF DIMENSION
```

22.1.5 Programming Examples

Example: Cylinder surface with Cycle 27

- Machine with B head and C table
- Cylinder centered on rotary table
- Preset is on the underside, in the center of the rotary table

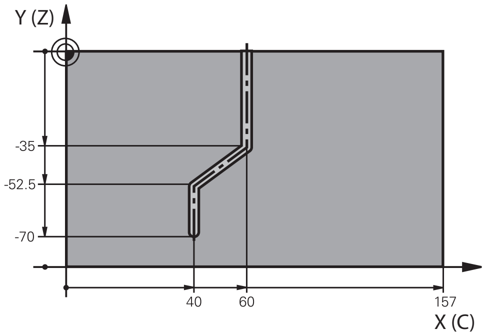


0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call (diameter: 7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-7 ;MILLING DEPTH ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q10=-4 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+250 ;FEED RATE F. ROUGHNG ~	
Q16=+25 ;RADIUS ~	
Q17=+1 ;TYPE OF DIMENSION	
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram
13 L X+40 Y-20 RL	; Rotary axis data in mm (Q17 = 1)
14 L X+50	
15 RND R7.5	
16 L Y-60	
17 RND R7.5	

18 L IX-20	
19 RND R7.5	
20 L Y-20	
21 RND R7.5	
22 L X+40 Y-20	
23 LBL 0	
24 END PGM 5 MM	

Example: Cylinder surface with Cycle 28

- Cylinder centered on rotary table
- Machine with B head and C table
- Preset is at the center of the rotary table
- Description of the path of the tool center in the contour subprogram



0 BEGIN PGM 4 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call, tool axis (Z), diameter (7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~	
Q1=-7 ;MILLING DEPTH ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q10=-4 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+250 ;FEED RATE F. ROUGHNG ~	
Q16=+25 ;RADIUS ~	
Q17=+1 ;TYPE OF DIMENSION ~	
Q20=+10 ;SLOT WIDTH ~	
Q21=+0.02 ;TOLERANCE	
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram, description of the path of the tool center
13 L X+60 Y+0 RL	; Rotary axis data in mm (Q17 = 1)
14 L Y-35	
15 L X+40 Y-52.5	

16 L X-70	
17 LBL 0	
18 END PGM 4 MM	

22.2 Working with the parallel axes U, V and W

22.2.1 Fundamentals

In addition to the main axes X, Y, and Z, the parallel axes U, V, and W, are available. A parallel axis is, for example, a spindle sleeve for boring so that smaller masses are moved on large machines.

Further information: "Programmable axes", Page 208

The control provides the following functions for machining with the parallel axes U, V and W:

- **FUNCTION PARAXCOMP**: Define behavior when positioning parallel axes

  Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1284
- **FUNCTION PARAXMODE**: Select three linear axes for machining

  Further information: "Select three linear axes for machining with FUNCTION PARAXMODE", Page 1289

If the machine manufacturer has already enabled the parallel axis in the configuration, the control takes this axis into account in the calculations, without you having to program **PARAXCOMP**. Since the control then continuously offsets the parallel axis, you can for example probe a workpiece even with any position of the W axis.

In this case, the control displays an icon in the **Positions** workspace.

Further information: "Positions workspace", Page 163

Please note that **PARAXCOMP OFF** does not deactivate the parallel axis in this case, but the control reactivates the standard configuration. The control deactivates automatic calculation only if you include the axis in the NC block (e.g. **PARAXCOMP OFF W**).

After the control has booted, the configuration defined by the machine manufacturer is in effect.

Requirements

- Machine with parallel axes
- Parallel axis functions activated by the machine manufacturer

  The machine manufacturer uses the optional machine parameter **parAxComp** (no. 300205) to define whether the parallel axis function is switched on by default.

22.2.2 Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP

Application

The **FUNCTION PARAXCOMP** function is used to define whether the control takes parallel axes into account in the traversing movements with the associated main axis.

Description of function

If the **FUNCTION PARAXCOMP** function is active, the control displays an icon in the **Positions** workspace. The icon for **FUNCTION PARAXMODE** may cover an active icon for **FUNCTION PARAXCOMP**.

Further information: "Positions workspace", Page 163

FUNCTION PARAXCOMP DISPLAY

Use the **PARAXCOMP DISPLAY** function to activate the display function for parallel axis movements. The control includes movements of the parallel axis in the position display of the associated main axis (sum display). Therefore, the position display of the main axis always displays the relative distance from the tool to the workpiece, regardless of whether you move the main axis or the parallel axis.

FUNCTION PARAXCOMP MOVE

The control uses the **PARAXCOMP MOVE** function to compensate for movements of a parallel axis by performing compensation movements in the associated main axis.

For example, if a parallel-axis movement is performed in the negative W-axis direction, the main axis Z is moved simultaneously in the positive direction by the same value. The relative distance from the tool to the workpiece remains the same. Application in gantry-type milling machines: Retract the spindle sleeve to move the cross beam down simultaneously.

FUNCTION PARAXCOMP OFF

Use the **PARAXCOMP OFF** function to switch off the **PARAXCOMP DISPLAY** and **PARAXCOMP MOVE** parallel axis functions.

The following actions cause the control to reset the **PARAXCOMP** parallel-axis function:

- Selection of NC program
- **PARAXCOMP OFF**

When **FUNCTION PARAXCOMP** is not active, the control does not display the corresponding icon and the additional information after the axis designations.

Input

11 FUNCTION PARAXCOMP MOVE W	; Compensate for movements of the W axis by means of a compensating movement in the Z axis

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION PARAXCOMP	Syntax initiator for the behavior when positioning parallel axes
DISPLAY, **MOVE** or **OFF**	Calculate the values of the parallel axis with the main axis, compensate for or do not take into account movements with the main axis
X, **Y**, **Z**, **U**, **V** or **W**	Affected axis Optional syntax element

Notes

- The **PARAXCOMP MOVE** function can be used only in connection with straight-line blocks (**L**).
- The control allows the use of one active **PARAXCOMP** function per axis only. If you define an axis both in **PARAXCOMP DISPLAY** and in **PARAXCOMP MOVE**, the last executed function will be active.
- Using offset values, you can define a parallel axis shift for the NC program (e.g., in the **W** axis). This allows machining of workpieces with different heights using the same NC program, for example.

  Further information: "Example", Page 1287

Notes about machine parameters

The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. For **FUNCTION PARAXCOMP**, the machine parameter applies to the parallel axes (**U_OFFS**, **V_OFFS**, and **W_OFFS**) only. If there are no offsets, the control behaves as described in the functional description.

Further information: "Description of function", Page 1285

Further information: "Basic transformation and offset", Page 2039

- If the machine parameter has not been defined for the parallel axis or has been defined with **FALSE**, the offset is only active in the parallel axis. The preset of the programmed parallel-axis coordinates is shifted by the offset value. The coordinates of the main axis still reference the workpiece preset.
- If the machine parameter for the parallel axis has been defined with **TRUE**, the offset will be active in the parallel and main axes. The presets of the programmed parallel and main axis coordinates are shifted by the offset value.

Example

This example shows the effect of the optional machine parameter **presetToAlignAxis** (no. 300203)

Machining is done on a gantry-type milling machine using a spindle sleeve as the **W** axis (parallel to the main **Z** axis). The **W_OFFS** column of the preset table contains the value **-10**. The Z value of the workpiece preset is located at the machine datum.

Further information: "Presets in the machine", Page 210

11 L Z+100 W+0 R0 FMAX M91	; Position the **Z** and **W** axes in the **M-CS** machine coordinate system
12 FUNCTION PARAX COMP DISPLAY W	; Activate the sum display
13 L Z+0 F1500	; Position the Z axis at 0
14 L W-20	; Move the W axis to working depth

In the first NC block, the control positions the **Z** and **W** axes relative to the machine datum, i.e. independent of the workpiece preset. In the **RFACTL** mode, the position display indicates the values **Z+100** and **W+0**. In the **ACTL.** mode, the control takes **W_OFFS** into account and displays the values **Z+100** and **W+10**.

Further information: "Position displays", Page 188

In NC block **12**, the control activates sum display for the **ACTL.** and **NOML.** modes of the position display. The control displays the movements of the W axis in the position display of the Z axis.

The result depends on the setting of the **presetToAlignAxis** machine parameter:

FALSE or not defined	TRUE
The control takes the offset into account in the W axis only. The value of the Z axis display remains unchanged.	The control takes the offset into account in the **W** and **Z** axes. The **ACTL.** display of the Z axis is changed by the offset value.
Position-display values: ■ **RFACTL** mode: **Z+100**, **W+0** ■ **ACTL.** mode: **Z+100**, **W+10**	Position-display values: ■ **RFACTL** mode: **Z+100**, **W+0** ■ **ACTL.** mode: **Z+110**, **W+10**

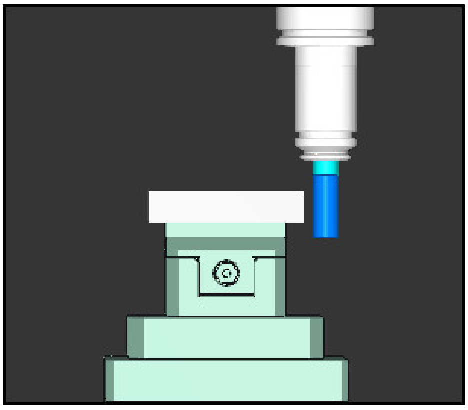
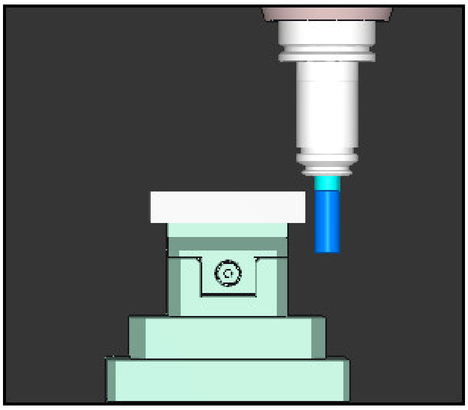
In NC block **13**, the control moves the Z axis to the programmed coordinate **0**.

The result depends on the setting of the **presetToAlignAxis** machine parameter:

FALSE or not defined	TRUE
The control moves the Z axis by 100 mm.	The coordinates of the Z axis reference the offset. To reach the programmed coordinate **0**, the axis must move by 110 mm.
Position-display values: ■ **RFACTL** mode: **Z+0**, **W+0** ■ **ACTL.** mode: **Z+0**, **W+10**	Position-display values: ■ **RFACTL** mode: **Z-10**, **W+0** ■ **ACTL.** mode: **Z+0**, **W+10**

In NC block **14**, the control moves the W axis to the programmed coordinate **-20**. The coordinates of the W axis reference the offset. To reach the programmed coordinate, the axis must move by 30 mm. Since the sum display has been activated, the control displays the movement in the **ACTL.** display of the Z axis as well.

The values in the position display depend on the setting of the **presetToAlignAxis** machine parameter:

FALSE or not defined	TRUE
Position-display values: ■ **RFACTL** mode: **Z+0**, **W-30** ■ **ACTL.** mode: **Z-30**, **W-20**	Position-display values: ■ **RFACTL** mode: **Z-10**, **W-30** ■ **ACTL.** mode: **Z-30**, **W-20**
 The tool tip is lower by the offset value than programmed in the NC program (**RFACTL W-30** instead of **W-20**).	 The tool tip is lower by the twice the offset value than programmed in the NC program (**RFACTL Z-10**, **W-30** instead of **Z+0**, **W-20**).

> If you only move the W axis while the **PARAXCOMP DISPLAY** function is active, the control takes the offset into account only once, independent of the setting of the **presetToAlignAxis** machine parameter.

22.2.3 Select three linear axes for machining with FUNCTION PARAXMODE

Application

Use the **PARAXMODE** function to define the axes the control is to use for machining. You program all traverses and contour descriptions in the main axes X, Y and Z, independent of your machine.

Requirement

- Parallel axis is calculated

  If your machine manufacturer has not yet activated the **PARAXCOMP** function as default, you must activate **PARAXCOMP** before you can work with **PARAXMODE**.

  Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1284

Description of function

If the **PARAXMODE** function is active, the control uses the axes defined in the function to execute the programmed traverses. If the control is to move the main axis deselected by **PARAXMODE**, you can identify this axis by additionally entering the **&** character. The **&** character then refers to the main axis.

Further information: "Moving the main axis and the parallel axis", Page 1290

Define three axes with the **PARAXMODE** function (e.g., **FUNCTION PARAXMODE X Y W**) to be used by the control for programmed traverses.

If the **FUNCTION PARAXMODE** function is active, the control displays an icon in the **Positions** workspace. The icon for **FUNCTION PARAXMODE** may cover an active icon for **FUNCTION PARAXCOMP**.

Further information: "Positions workspace", Page 163

FUNCTION PARAXMODE OFF

Use the **PARAXCOMP OFF** function to deactivate the parallel-axis function. The control then uses the main axes defined by the machine manufacturer.

The control resets the **PARAXMODE ON** parallel-axis function via the following functions:

- Selection of an NC program
- End of program
- **M2** and **M30**
- **PARAXMODE OFF**

Input

11 FUNCTION PARAX MODE X Y W	; Execute programmed traversing movements with axes **X**, **Y** and **W.**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION PARAX MODE	Syntax initiator for axis selection for machining
OFF	Deactivate the parallel axis function Optional syntax element
X, **Y**, **Z**, **U**, **V** or **W**	Three axes for machining Only for **FUNCTION PARAX MODE**

Moving the main axis and the parallel axis

If the **PARAXMODE** function is active, you can traverse the deselected main axis with the **&** character within the straight line **L**.

Further information: "Straight line L", Page 326

To traverse a deselected main axis:

- Select **L**
- Define coordinates
- Select deselected main axis (e.g., **&Z**)
- Enter a value
- Define the radius compensation, if necessary
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary
- Confirm your input

Notes

- You must deactivate the parallel-axis functions before switching the machine kinematics.
- In order for the control to offset the main axis deselected with **PARAXMODE**, enable the **PARAXCOMP** function for this axis.
- Additional positioning of a main axis with the **&** command is done in the REF system. If you have set the position display to display ACTUAL values, this movement will not be shown. If necessary, switch the position display to REF values.

  Further information: "Position displays", Page 188

Notes about machine parameters

- You can deactivate the programming of parallel axes with the machine parameter **noParaxMode** (no. 105413).
- Your machine manufacturer will define the calculation of possible offset values (X_OFFS, Y_OFFS and Z_OFFS from the preset table) for the axes positioned with the **&** operator in the **presetToAlignAxis** machine parameter (no. 300203).
  - If the machine parameter has not been defined for the main axis or has been defined with **FALSE**, the offset only applies to the axis programmed with **&**. The coordinates of the parallel axis still reference the workpiece preset. Despite the offset, the parallel axis will move to the programmed coordinates.
  - If the machine parameter for the main axis has been defined with **TRUE**, the offset applies to the main axis and the parallel axis. The presets of the main and parallel axis coordinates are shifted by the offset value.

22.2.4 Parallel axes in conjunction with machining cycles

You can also use most machining cycles of the control with parallel axes.

Further information: "Machining Cycles", Page 475

You cannot use the following cycles with parallel axes:

- Cycle **285 DEFINE GEAR** (option 157)
- Cycle **286 GEAR HOBBING** (option 157)
- Cycle **287 GEAR SKIVING** (option 157)
- Touch probe cycles

22.2.5 Example

Drilling is carried out with the W axis in the following NC program:

0 BEGIN PGM PAR 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 5 Z S2222	; Call the tool in the tool axis **Z**
4 L Z+100 R0 FMAX M3	; Position the main axis
5 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	
6 FUNCTION PARAXCOMP DISPLAY Z	; Activate display compensation
7 FUNCTION PARAXMODE X Y W	; Positive axis selection
8 L X+50 Y+50 R0 FMAX M99	; The parallel axis **W** executes the infeed
9 FUNCTION PARAXMODE OFF	; Restore the standard configuration
10 L M30	
11 END PGM PAR MM	

22.3 Using a facing slide with FACING HEAD POS (option 50)

Application

With a facing slide, also called boring head, you can perform almost all turning operations with fewer different tools. The slide position of the facing slide in the X direction can be programmed. On the facing slide you mount, for example, a longitudinal turning tool that you call with a TOOL CALL block.

Related topics

- Machining with parallel axes **U**, **V** and **W**

  Further information: "Working with the parallel axes U, V and W", Page 1284

Requirements

- Combined milling/turning (software option 50)
- Control prepared by the machine manufacturer

  The machine manufacturer must take the facing head into account in the kinematics.
- Kinematics with facing head activated

  Further information: "Switching the operating mode with FUNCTION MODE", Page 234
- Workpiece datum in the working plane is at the center of the rotationally symmetrical contour

  With a facing slide, the workpiece datum must not be in the center of the rotary table, because the tool spindle rotates.

  Further information: "Datum shift with TRANS DATUM", Page 1046

Description of function

Refer to your machine manual.

The machine manufacturer can provide customized cycles for working with a facing slide. The standard functionality is described below.

The facing slide is defined as a turning tool.

Further information: "Turning tool table toolturn.trn (option 50)", Page 2006

Please note for tool calls:

- **TOOL CALL** block without tool axis
- Cutting speed and spindle speed with **TURNDATA SPIN**
- Switch the spindle on with **M3** or **M4**

Machining also works with a tilted working plane and on workpieces that are not rotationally symmetric.

If you move with the facing slide without the **FACING HEAD POS** function, you must program the motions of the facing slide in the U axis (e.g., in the **Manual operation** application). If the **FACING HEAD POS** function is active, program the facing slide movements in the X axis.

When you activate the facing slide, the control automatically positions itself at the workpiece datum in **X** and **Y**. To avoid collisions, you can define a safe height using the **HEIGHT** syntax element.

The facing slide is deactivated with the **FUNCTION FACING HEAD** function.

Input

Activating the facing slide

11 FACING HEAD POS HEIGHT+100 FMAX	; Activate facing slide and move with rapid traverse to safe height **Z**+100

The NC function includes the following syntax elements:

Syntax element	Meaning
FACING HEAD POS	Activate the syntax initiator for the facing slide
HEIGHT	Safe height in the tool axis Optional syntax element
F or **FMAX**	Approach safe height with defined feed rate or rapid traverse Optional syntax element
M	Additional function Optional syntax element

Deactivating the facing slide

11 FUNCTION FACING HEAD OFF	; Deactivate facing slide

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FACING HEAD OFF	Deactivate the syntax initiator for the facing slide

Notes

NOTICE

Caution: Danger to the tool and workpiece!

For a facing slide to be used, a kinematic model prepared by the machine manufacturer must be selected by means of the function **FUNCTION MODE TURN**. In this kinematic model, the control implements the programmed X-axis movements of the facing slide as U-axis movements when the **FACING HEAD** function is active. When the **FACING HEAD** function is not active and in **Manual operation** operating mode, this automated implementation does not take place. As a result, **X** axis movements (programmed or via axis key) will be performed in the X axis. In this case, the facing slide has to be moved with the U axis. There is a danger of collision during retraction or manual movements!

- Position the facing slide at its home position while the **FACING HEAD POS** function is active
- Retract the facing slide while the **FACING HEAD POS** function is active
- In the **Manual operation** operating mode, move the facing slide with the **U** axis key.
- As the **Tilt working plane** function can be used, pay attention to the 3-D ROT status

- To set a spindle-speed limitation, you can use the **NMAX** value from the tool table as well as the **SMAX** value from **FUNCTION TURNDATA SPIN**.
- The following constraints apply to the use of a facing slide:
  - Miscellaneous functions **M91** and **M92** cannot be used
  - Retraction with **M140** is not possible
  - **TCPM** or **M128** are not possible (option 9)
  - **DCM** collision monitoring cannot be used (option 40)
  - Cycles **800**, **801**, and **880** cannot be used
  - Cycles **286** and **287** cannot be used (option 157)
- If you are using the facing slide in the tilted working plane, please note the following:
  - The control calculates the tilted working plane as in milling mode. The **COORD ROT** and **TABLE ROT** functions, as well as **SYM (SEQ)**, reference the XY plane.

    Further information: "Tilting solution", Page 1091
  - HEIDENHAIN recommends selecting the **TURN** positioning behavior. The **MOVE** positioning behavior is not the best option in combination with the facing slide.

    Further information: "Rotary axis positioning", Page 1088

Notes about machine parameters

The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. If **FACING HEAD POS** is used, the machine parameter applies to the parallel axis (**U** axis) only (**U_OFFS**).

Further information: "Basic transformation and offset", Page 2039

- If the machine parameter has not been defined or has been set to **FALSE**, the control does not take the offset into account during machining.
- If the machine parameter axis has been set to **TRUE**, the offset can be used to compensate a facing slide offset. If you are using a facing slide with multiple tool clamp options, set the offset for the current clamping position. This ensures that you can run NC programs independent of the tool clamping position.

22.4 Machining with polar kinematics with FUNCTION POLARKIN

Application

In a polar kinematic model, the path contours of the working plane are performed by one linear axis and one rotary axis instead of by two linear principal axes. The working plane is defined by the linear principal axis and the rotary axis while the working space is defined by these two axes and the infeed axis.

On milling machines, various linear principal axes can be replaced with suitable rotary axes. For example on large machines, polar kinematics enable you to machine much larger surfaces than with only the principal axes.

On turning and grinding machines that have only two linear principal axes, polar kinematics enable milling operations to be performed on the front face.

Requirements

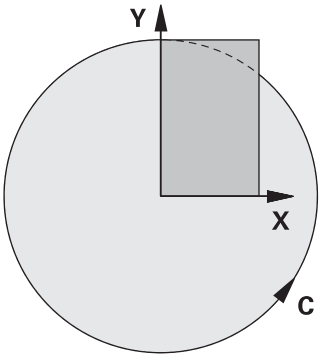
- Machine with at least one rotary axis

  The polar rotary axis must be installed onto the table side so that it is opposite the selected linear axes and must be configured as a modulo axis. Thus, the linear axes must not be positioned between the rotary axis and the table. The maximum range of traverse of the rotary axis is limited by the software limit switches if necessary.
- **PARAXCOMP DISPLAY** function programmed with at least the main axes **X**, **Y** and **Z**.

  HEIDENHAIN recommends defining all of the available axes within the **PARAXCOMP DISPLAY** function.

  Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1284

Description of function

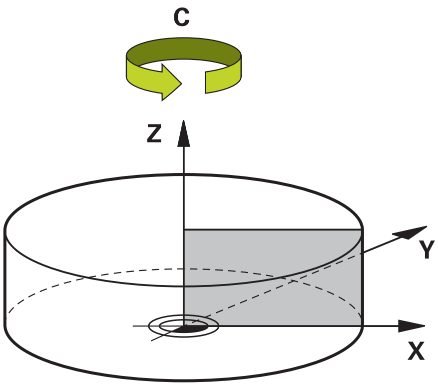


When the polar kinematics are active, the control displays an icon in the **Positions** workspace. This icon covers the icon for the **PARAXCOMP DISPLAY** function.

Use the **POLARKIN AXES** function to activate the polar kinematics. The axis data define the radial axis, the infeed axis, and the polar axis. The **MODE** data influence the positioning behavior, whereas the **POLE** data define the machining at the pole. The pole is the center of rotation of the rotary axis in this case.

Notes on the axes to be selected:

- The first linear axis must be radial to the rotary axis.
- The second linear axis defines the infeed axis and must be parallel to the rotary axis.
- The rotary axis defines the polar axis and is defined last.
- Any available modulo axis that is installed at the table opposite to the selected linear axes can be used as the rotary axis.
- The two selected linear axes thus span a plane that also includes the rotary axis.



The following scenarios lead to deactivation of the polar kinematics:

- Execution of the **POLARKIN OFF** function
- Selection of an NC program
- Reaching the end of the NC program
- Abortion of the NC program
- Selecting a kinematic model
- Restarting the control

MODE options

The control provides the following options for positioning behavior:

MODE options:

Syntax	Function
POS	Seen from the center of rotation, the control performs machining in the positive direction of the radial axis. The radial axis must be prepositioned correspondingly.
NEG	Seen from the center of rotation, the control performs machining in the negative direction of the radial axis. The radial axis must be prepositioned correspondingly.
KEEP	The control remains with the radial axis on that side of the center of rotation on which the axis was positioned when the function was activated. If the radial axis is positioned at the center of rotation upon switch-on, **POS** applies.
ANG	The control remains with the radial axis on that side of the center of rotation on which the axis was positioned when the function was activated. If you set **POLE** to **ALLOWED**, positioning through the pole is possible. The pole side is changed and a 180-degree rotation of the rotary axis is prevented.

POLE options

The control provides the following options for machining at the pole:

POLE options:

Syntax	Function
ALLOWED	The control permits machining operations at the pole
SKIPPED	The control prevents machining operations at the pole

> The disabled area corresponds to a circular surface with a radius of 0.001 mm (1 µm) around the pole.

Input

11 FUNCTION POLARKIN AXES X Z C MODE: KEEP POLE: ALLOWED	; Activate polar kinematics with axes **X**, **Z** and **C**.

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION POLARKIN	Syntax initiator for polar kinematics
AXES or **OFF**	Activate or deactivate polar kinematics
X, **Y**, **Z**, **U**, **V**, **A**, **B**, **C**	Selection of two linear axes and one rotary axis Only when **AXES** is selected Other possibilities might be available, depending on the machine.
MODE:	Selection of the positioning behavior **Further information:** "MODE options", Page 1297 Only when **AXES** is selected
POLE:	Selection of machining in the pole **Further information:** "POLE options", Page 1297 Only when **AXES** is selected

Notes

- The principal axes X, Y, and Z as well as the possible parallel axes U, V, and W can be used as radial axes or infeed axes.
- Position the linear axis that will not be included in the polar kinematics to the coordinate of the pole, before the **POLARKIN** function. Otherwise, a non-machinable area with a radius that corresponds to at least the value of the deselected linear axis would result.
- Avoid performing machining operations at the pole or near the pole, because feed-rate variations may occur in this area. For this reason, ideally use the following **POLE** option: **SKIPPED**.
- Polar kinematics cannot be combined with the following functions:
  - Traverses with **M91**

    Further information: "Traversing in the machine coordinate system M-CS with M91", Page 1322
  - Tilting the working plane (option 8)
  - **FUNCTION TCPM** or **M128** (option 9)
- Note that the traversing range of the axes may be limited.

  Further information: "Notes on software limit switches for modulo axes", Page 1311

  Further information: "Traverse limits", Page 2104

Notes about machine parameters

- The machine manufacturer uses the optional machine parameter **kindOfPref** (no. 202301) to define the behavior of the control when the path of the tool center point passes through the polar axis.
- The machine manufacturer uses the optional machine parameter **preset-ToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. For **FUNCTION POLARKIN**, the machine parameter applies only to the rotary axis that rotates about the tool axis (in most cases **C_OFFS**).

  Further information: "Comparison of offset and 3D basic rotation", Page 1580

  - If the machine parameter axis has not been defined or has been set to **TRUE**, the offset can be used to compensate a misalignment of the workpiece in the plane. The offset affects the orientation of the workpiece coordinate system **W-CS**.

    Further information: "Workpiece coordinate system W-CS", Page 1016
  - If the machine parameter axis has been defined with **FALSE**, the offset cannot be used to compensate a misalignment of the workpiece in the plane. The control will not take the offset into account when executing the commands.

22.4.1 Example: SL cycles in the polar kinematics

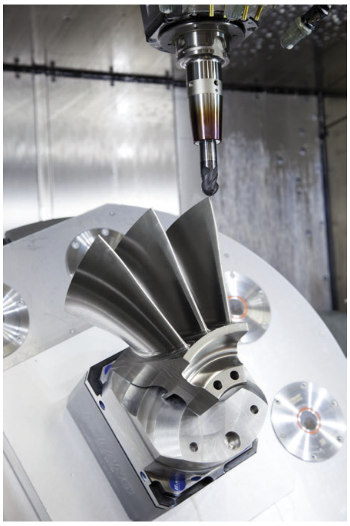
0 BEGIN PGM POLARKIN_SL MM	
1 BLK FORM 0.1 Z X-100 Y-100 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 2 Z S2000 F750	
4 FUNCTION PARAXCOMP DISPLAY X Y Z	; Activate **PARAXCOMP DISPLAY**
5 L X+0 Y+0.0011 Z+10 A+0 C+0 FMAX M3	; Pre-position outside the disabled pole area
6 POLARKIN AXES Y Z C MODE:KEEP POLE:SKIPPED	; Activate **POLARKIN**
* - ...	; Datum shift in polar kinematics
9 TRANS DATUM AXIS X+50 Y+50 Z+0	
10 CYCL DEF 7.3 Z+0	
11 CYCL DEF 14.0 CONTOUR	
12 CYCL DEF 14.1 CONTOUR LABEL2	
13 CYCL DEF 20 CONTOUR DATA	
Q1=-10 ;MILLING DEPTH	
Q2=+1 ;TOOL PATH OVERLAP	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=+2 ;SET-UP CLEARANCE	
Q7=+50 ;CLEARANCE HEIGHT	
Q8=+0 ;ROUNDING RADIUS	
Q9=+1 ;ROTATIONAL DIRECTION	
14 CYCL DEF 22 ROUGH-OUT	
Q10=-5 ;PLUNGING DEPTH	
Q11=+150 ;FEED RATE FOR PLNGNG	
Q12=+500 ;FEED RATE F. ROUGHNG	
Q18=+0 ;COARSE ROUGHING TOOL	
Q19=+0 ;FEED RATE FOR RECIP.	
Q208=+99999 ;RETRACTION FEED RATE	
Q401=+100 ;FEED RATE FACTOR	
Q404=+0 ;FINE ROUGH STRATEGY	
15 M99	
16 CYCL DEF 7.0 DATUM SHIFT	
17 CYCL DEF 7.1 X+0	
18 CYCL DEF 7.2 Y+0	
19 CYCL DEF 7.3 Z+0	
20 POLARKIN OFF	; Deactivate **POLARKIN**
21 FUNCTION PARAXCOMP OFF X Y Z	; Deactivate **PARAXCOMP DISPLAY**
22 L X+0 Y+0 Z+10 A+0 C+0 FMAX	
23 L M30	
24 LBL 2	

25 L X-20 Y-20 RR	
26 L X+0 Y+20	
27 L X+20 Y-20	
28 L X-20 Y-20	
29 LBL 0	
30 END PGM POLARKIN_SL MM	

22.5 CAM-generated NC programs

Application

CAM-generated NC programs are created externally of the control using CAM systems. In combination with 5-axis simultaneous machining and free-form surfaces, CAM systems provide a convenient solution, which in some cases may be the only solution possible.

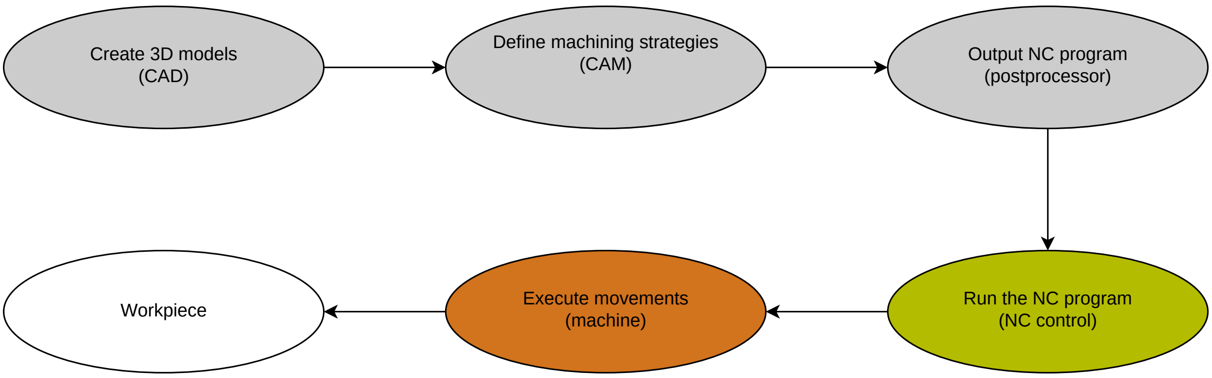


For CAM-generated NC programs to be able to use the full performance potential of the control and to provide you with such options as intervention and correction, certain requirements must be met.

CAM-generated NC programs must meet the same requirements as manually created NC programs. In addition, other requirements arise from the process chain.

Further information: "Process steps", Page 1306

The process chain specifies the path from a design to the finished workpiece.



Related topics

- Using 3D data directly at the control
  Further information: "Opening CAD Files with the CAD-Viewer", Page 1455
- Programming graphically
  Further information: "Graphical Programming", Page 1437

22.5.1 Output formats of NC programs

Output in HEIDENHAIN Klartext format

If you output the NC program in Klartext, you have the following options:

- 3-axis output
- Output with up to five axes, without **M128** or **FUNCTION TCPM**
- Output with up to five axes, with **M128** or **FUNCTION TCPM**

> Requirements for 5-axis machining:
> - Machine with rotary axes
> - Advanced Functions Set 1 (option 8)
> - Advanced Functions Set 2 (option 9) for **M128** or **FUNCTION TCPM**

If the machine kinematics and exact tool data are available to the CAM system, you can output 5-axis NC programs without **M128** or **FUNCTION TCPM**. The programmed feed rate is calculated for all axis components per NC block, which can result in different cutting speeds.

An NC program with **M128** or **FUNCTION TCPM** is machine-neutral and more flexible, since the control takes over the kinematics calculation and uses the tool data from the tool management. The programmed feed rate acts on the tool location point.

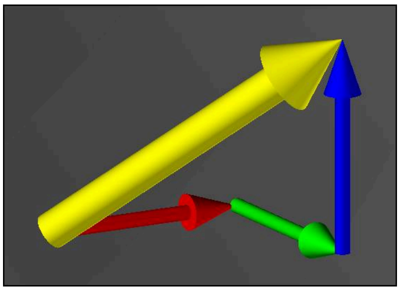
Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

Further information: "Presets on the tool", Page 271

Examples

11 L X+88 Y+23.5375 Z-8.3 R0 F5000	; 3-axis
11 L X+88 Y+23.5375 Z-8.3 A+1.5 C+45 R0 F5000	; 5-axis without **M128**
11 L X+88 Y+23.5375 Z-8.3 A+1.5 C+45 R0 F5000 M128	; 5-axis with **M128**

Output with vectors



From the point of view of physics and geometry, a vector is a directed variable that describes a direction and a length.

When outputting with vectors, the control requires at least one normalized vector that specifies the direction of the surface normals or the tool position. Optionally, the NC block contains both vectors.

A normalized vector is a vector with the value 1. The vector amount corresponds to the root of the sum of the squares of its components.

$$\sqrt{NX^2 + NY^2 + NZ^2} = 1$$

> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (option 8)
> - Advanced Functions Set 2 (option 9)

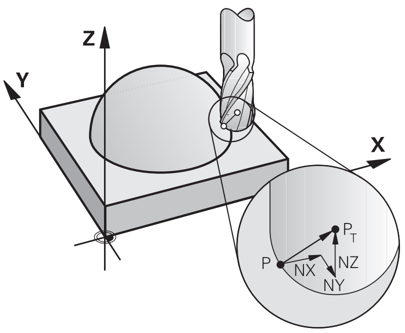
> You can only use the output with vectors in milling mode.
> **Further information:** "Switching the operating mode with FUNCTION MODE", Page 234

> Vector output with the direction of the surface normals is required for using 3D tool radius compensation depending on the tool's contact angle (option 92).
> **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1140

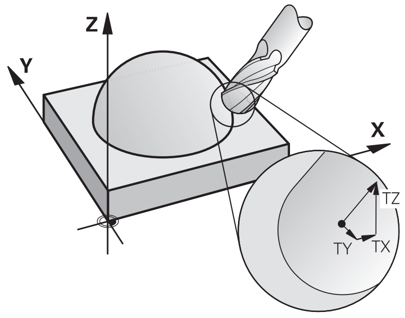
Examples

11 LN X0.499 Y-3.112 Z-17.105 NX0.2196165 NY-0.1369522 NZ0.9659258	; 3-axis with surface normal vector, without tool orientation
11 LN X0.499 Y-3.112 Z-17.105 NX0.2196165 NY-0.1369522 NZ0.9659258 TX+0.0078922 TY-0.8764339 TZ+0.2590319 M128	; 5-axis with M128, surface normal vector and tool orientation

Structure of an NC block with vectors



Surface normal vector perpendicular to the contour



Tool direction vector

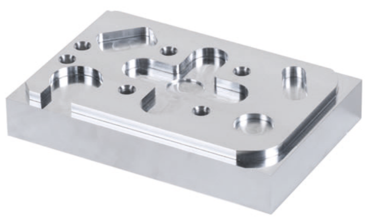
Example

11 LN X+0.499 Y-3.112 Z-17.105 NX0 NY0 NZ1 TX+0.0078922 TY-0.8764339 TZ+0.2590319	; Straight line **LN** with surface normal vector and tool orientation

Syntax element	Meaning
LN	Straight line **LN** with surface normal vector
X Y Z	Target coordinates
NX NY NZ	Components of the surface normal vector
TX TY TZ	Components of the tool direction vector

22.5.2 Types of machining according to number of axes

3-axis machining



If only the linear axes **X**, **Y** and **Z** are required for machining a workpiece, 3-axis machining takes place.

3+2-axis machining

If tilting of the working plane is required for machining a workpiece, 3+2-axis machining takes place.

> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (option 8)

Inclined machining

For inclined machining, also referred to as inclined-tool machining, the tool is positioned at a user-defined angle to the working plane. The orientation of the working plane coordinate system **WPL-CS** is not changed, but only the position of the rotary axes and therefore the tool position. The control is able to compensate for the offset that is created in the linear axes.

Inclined machining is used in conjunction with undercuts and short tool clamping lengths.

> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (option 8)
> - Advanced Functions Set 2 (option 9)

5-axis machining

In 5-axis machining, also referred to as 5-axis simultaneous machining, the machine moves five axes at the same time. For free-form surfaces, this means that the tool can always be oriented perfectly with respect to the workpiece surface.

> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (option 8)
> - Advanced Functions Set 2 (option 9)
>
> 5-axis machining is not possible with the export version of the control.

22.5.3 Process steps

CAD

Application

Using CAD systems, designers create the 3D models of the required workpieces. Incorrect CAD data has a negative impact on the entire process chain, including the quality of the workpiece.

Notes

- In 3D models, avoid open or overlapping faces and unnecessary points. If possible, use the check functions of the CAD system.
- Design or save the 3D models based on the center of tolerance and not the nominal dimensions.

> Support manufacturing with additional files:
> - Provide 3D models in STL format. The control-internal simulation can use the CAD data as blank and finished parts, for example. Additional models of tool and workholding equipment are important in conjunction with collision testing (option 40).
> - Provide drawings with the dimensions to be checked. The file type of the drawings is not important in this respect, since the control can also open files such as PDFs, and therefore supports paperless production.

Definition

Abbreviation	Definition
CAD (computer-aided design)	Computer-aided design

CAM and postprocessor

Application

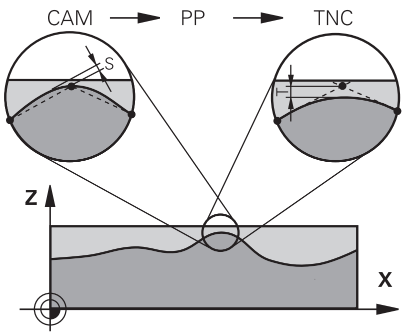
Using machining strategies within the CAM systems, CAM programmers create machine-independent and control-independent NC programs based on the CAD data.

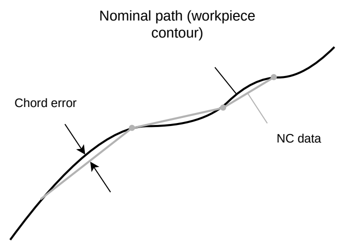
With the aid of the postprocessor, the NC programs are ultimately output specific to machine and control.

Notes on CAD data

- Avoid quality losses due to unsuitable transfer formats. Integrated CAM systems with manufacturer-specific interfaces work in some cases without loss.
- Take advantage of the available accuracy of the CAD data obtained. A geometry or model error of less than 1 µm is recommended for finishing large radii.

Notes on chord errors and Cycle 32 TOLERANCE





- In roughing, the focus is on the processing speed.

  The sum of the chord error and the tolerance **T** in Cycle **32 TOLERANCE** must be smaller than the contour allowance, otherwise contour violations may occur.

Chord error in CAM system	0.004 mm to 0.015 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.05 mm to 0.3 mm

- When finishing with the aim of high accuracy, the values must provide the required data density.

Chord error in CAM system	0.001 mm to 0.004 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.002 mm to 0.006 mm

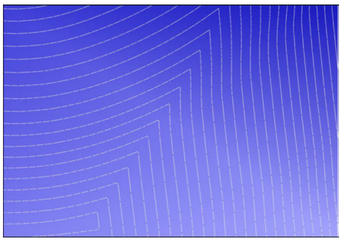
- When finishing with the aim of a high surface quality, the values must allow smoothing of the contour.

Chord error in CAM system	0.001 mm to 0.005 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.010 mm to 0.020 mm

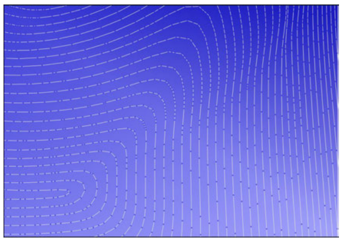
Further information: "Cycle 32 TOLERANCE ", Page 1213

Notes on control-optimized NC output

- Prevent rounding errors by outputting axis positions with at least four decimal places. For optical components and workpieces with large radii (small curves), at least five decimal places are recommended. The output of surface normal vectors (for straight lines **LN**) requires at least seven decimal places.
- You can prevent the cumulation of tolerances by outputting absolute instead of incremental coordinate values for successive positioning blocks.
- If possible, output positioning blocks as arcs. The control calculates circles more accurately internally.
- Avoid repetitions of identical positions, feed specifications and additional functions (e.g., **M3**).
- Output Cycle **32 TOLERANCE** again only when changing settings.
- Make sure that corners (curvature transitions) are precisely defined by an NC block.
- The feed rate fluctuates strongly if the tool path is output with strong changes in direction. If possible, round the tool paths.



Tool paths with strong changes in direction at transitions



Tool paths with rounded transitions

- Do not use intermediate or interpolation points for straight paths. These points are generated, for example, by a constant point output.
- Prevent patterns on the workpiece surface by avoiding exactly synchronous point distribution on surfaces with even curvature.
- Use suitable point distances for the workpiece and the machining step. Possible starting values are between 0.25 mm and 0.5 mm. Values greater than 2.5 mm are not recommended, even with high machining feed rates.
- Prevent mispositioning by outputting the **PLANE** functions (option 8) with **MOVE** or **TURN** without separate positioning blocks. If you output **STAY** and position the rotary axes separately, use variables **Q120** to **Q122** instead of fixed axis values.

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 1054
- Prevent strong feed breaks at the tool location point by avoiding an unfavorable relationship between linear and rotary axis motion. A significant change in the tool adjustment angle with a slight change in the position of the tool is a problem, for example. Take into account the different speeds of the axes involved.
- If the machine moves five axes simultaneously, the kinematic errors of the axes may multiply. Use as few axes as possible simultaneously.
- Avoid unnecessary feed rate limits that you can define within **M128** or the **FUNCTION TCPM** (option 9) function for compensation movements.

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
- Take into account the machine-specific behavior of rotary axes.

  Further information: "Notes on software limit switches for modulo axes", Page 1311

Notes on tools

- A ball-nose cutter, a CAM output to the tool center point and a high rotational axis tolerance **TA** (1° to 3°) in cycle **32 TOLERANCE** enable uniform feed paths.
- Ball-nose or toroidal milling cutter and a CAM output relative to the tool tip require low rotational axis tolerances **TA** (approx. 0.1°) in Cycle **32 TOLERANCE**. Contour violations are more likely to occur at higher values. The extent of the contour violations depends on factors such as the tool position, the tool radius and the depth of engagement.

Further information: "Presets on the tool", Page 271

Notes on user-friendly NC outputs

- Facilitate the easy adaptation of NC programs by using the machining and touch probe cycles of the control.
- Facilitate both the adaptation options and the overview by defining feed rates centrally using variables. It is preferable to use freely usable variables (e.g., **QL** parameters).

  Further information: "Variables: Q, QL, QR and QS parameters", Page 1362
- Provide a better overview by structuring the NC programs. One method is to use subprograms within the NC programs. If possible, divide larger projects into multiple separate NC programs.

  Further information: "Programming Techniques", Page 383
- Support correction options by outputting contours with tool radius correction.

  Further information: "Tool radius compensation", Page 1114
- Use structure items to enable fast navigation within the NC programs.

  Further information: "Structuring of NC programs", Page 1514
- Use comments to communicate important information about the NC program.

  Further information: "Adding comments", Page 1512

NC control and machine

Application

The control uses the points defined in the NC program to calculate the motions of each machine axis as well as the required velocity profiles. Control-internal filter functions then process and smooth the contour so that the control does not exceed the maximum permissible path deviation.

The motions and velocity profiles calculated are implemented as movements of the tool by the machine's drive system.

You can use various intervention and correction options to optimize machining.

Notes on the use of CAM-generated NC programs

- The simulation of machine and control-independent NC data within the CAM systems can deviate from the actual machining. Check the CAM-generated NC programs using the control-internal simulation.

  Further information: "Simulation Workspace", Page 1535
- Take into account the machine-specific behavior of rotary axes.

  Further information: "Notes on software limit switches for modulo axes", Page 1311
- Make sure that the required tools are available and that the remaining service life is sufficient.

  Further information: "Tool usage test", Page 312
- If necessary, change the values in Cycle **32 TOLERANCE** depending on the chord error and the dynamic response of the machine.

  Further information: "Cycle 32 TOLERANCE ", Page 1213

  > Refer to your machine manual.
  >
  > Some machine manufacturers provide an additional cycle for adapting the behavior of the machine to the respective machining operation (e.g. Cycle **332 Tuning**). Cycle **332** can be used to modify filter settings, acceleration settings and jerk settings.

- If the CAM-generated NC program contains normalized vectors, you can also correct tools three-dimensionally.

  Further information: "Output formats of NC programs", Page 1302

  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1140
- Software options enable further optimizations.

  Further information: "Functions and function packages", Page 1313

  Further information: "Software options", Page 94

Notes on software limit switches for modulo axes

> The following information on software limit switches for modulo axes also applies to traversing limits.
>
> **Further information:** "Traverse limits", Page 2104

The following general conditions apply to software limit switches for modulo axes:

- The lower limit is greater than –360° and less than +360°.
- The upper limit is not negative and less than +360°.
- The lower limit is not greater than the upper limit.
- The lower and upper limits are less than 360° apart.

If the general conditions are not met, the control cannot move the modulo axis and issues an error message.

If the target position or a position equivalent to it is within the permitted range, movement is permitted with active modulo limit switches. The direction of motion is determined automatically, as only one of the positions can be approached at any one time. Please note the following examples!

Equivalent positions differ by an offset of n x 360° from the target position. The factor n corresponds to any integer.

Example

11 L C+0 R0 F5000	; Limit switches –80° and +80°
12 L C+320	; Target position –40°

The control positions the modulo axis between the active limit switches to the position –40°, which is equivalent to 320°.

Example

11 L C-100 R0 F5000	; Limit switches –90° and +90°
12 L IC+15	; Target position –85°

The control executes the traversing motion because the target position lies within the permitted range. The control positions the axis in the direction of the nearest limit switch.

Example

11 L C-100 R0 F5000	; Limit switches –90° and +90°
12 L IC-15	; Error message

The control issues an error message because the target position is outside the permitted range.

Examples

11 L C+180 R0 F5000	; Limit switches –90° and +90°
12 L C-360	; Target position 0°: Also applies for a multiple of 360°, e.g. 720°

11 L C+180 R0 F5000	; Limit switches –90° and +90°
12 L C+360	; Target position 360°: Also applies for a multiple of 360°, e.g. 720°

If the axis is exactly in the middle of the prohibited area, the distance to both limit switches is identical. In this case, the control can move the axis in both directions.

If the positioning block results in two equivalent target positions in the permitted range, the control positions itself along the shorter path. If both equivalent target positions are 180° away, the control selects the direction of motion according to the programmed algebraic sign.

Definitions

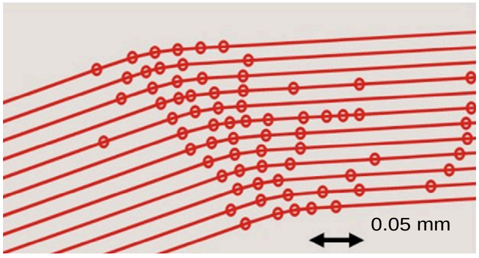
Modulo axis
Modulo axes are axes whose encoder only returns values between 0° and 359.9999°. If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

Rollover axis
Rollover axes are rotary axes that can perform several or any number of revolutions. The machine manufacturer must configure a rollover axis as a modulo axis.

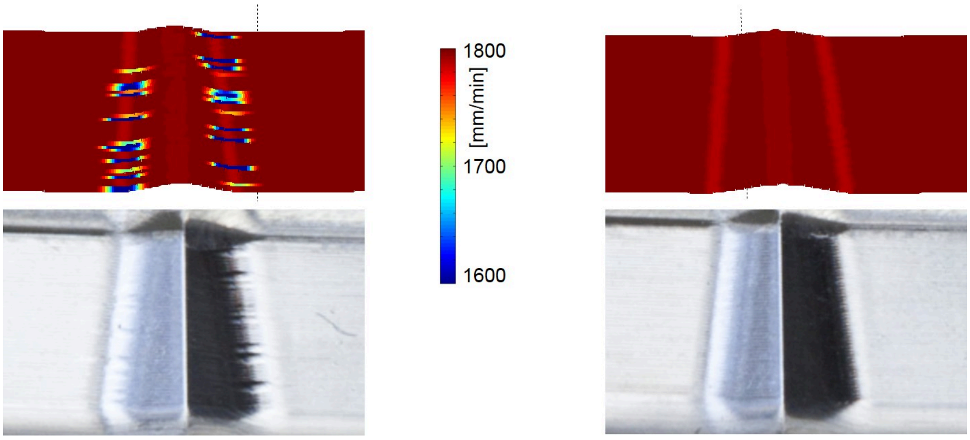
Modulo counting method
The position display of a rotary axis with the modulo counting method is between 0° and 359.9999°. If the value exceeds 359.9999°, the display starts over at 0°.

22.5.4 Functions and function packages

ADP motion control



Distribution of points



Comparison without and with ADP

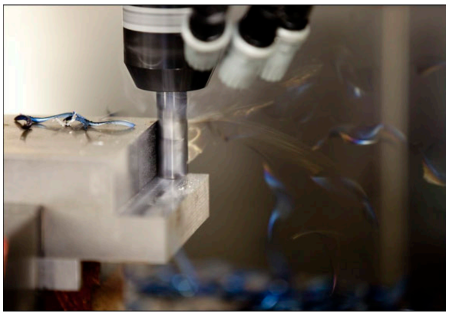
CAM-generated NC programs with an insufficient resolution and variable point density in adjacent paths can lead to feed rate fluctuations and errors on the workpiece surface.

The Advanced Dynamic Prediction (ADP) function extends the prediction of the permissible maximum feed rate profile and optimizes the motion control of the axes involved during milling. This means that you can achieve a high surface quality with a short machining time and reduce the reworking effort.

The most important benefits of ADP at a glance:

- With bidirectional milling, the forward and reverse paths have symmetrical feed behavior.
- Tool paths adjacent to one another have uniform feed paths.
- Negative effects associated with typical problems of CAM-generated NC programs are compensated for or mitigated, e.g.:
  - Short stair-like steps
  - Rough chord tolerances
  - Strong rounded block end point coordinates
- Even under difficult conditions, the control precisely complies with the dynamic parameters.

Dynamic Efficiency



The Dynamic Efficiency package of functions enables you to increase process reliability in heavy machining and roughing in order to improve efficiency.

Dynamic Efficiency includes the following software features:

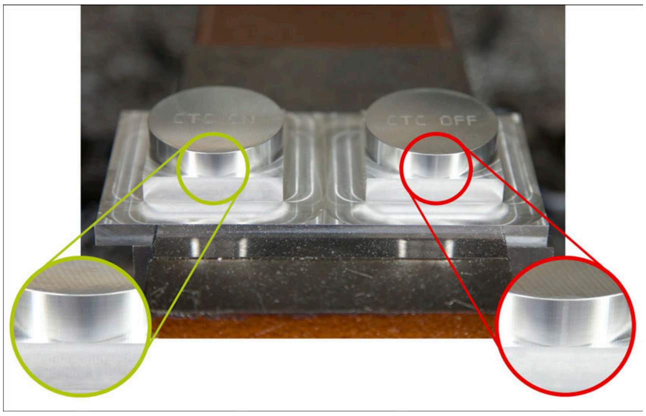
- Active Chatter Control (ACC, option 145)
- Adaptive Feed Control (AFC, option 45)
- Cycles for trochoidal milling (option 167)

Using Dynamic Efficiency offers the following advantages:

- ACC, AFC and trochoidal milling reduce machining time by increasing the material removal rate.
- AFC enables tool monitoring and thus increases process reliability.
- ACC and trochoidal milling extend the tool life.

You can find more information in the brochure titled **Options and Accessories.**

Dynamic Precision



The Dynamic Precision package of functions enables you to machine quickly and accurately, and with high surface quality.

Dynamic Precision includes the following software functions:

- Cross Talk Compensation (CTC, option 141)
- Position Adaptive Control (PAC, option 142)
- Load Adaptive Control (LAC, option 143)
- Motion Adaptive Control (MAC, option 144)
- Active Vibration Damping (AVD, option 146)

The functions each provide decisive improvements. They can be combined and also mutually complement each other:

- CTC increases the accuracy in the acceleration phases.
- AVD enables better surfaces.
- CTC and AVD result in fast and accurate processing.
- PAC leads to increased contour constancy.
- LAC keeps accuracy constant, even with variable load.
- MAC reduces vibrations and increases the maximum acceleration for rapid traverse movements.

> You can find more information in the brochure titled **Options and Accessories.**

23

Miscellaneous Functions

23.1 Miscellaneous functions M and the STOP function

Application

Use miscellaneous functions to activate or deactivate functions of the control and to influence the behavior of the control.

Description of function

You can define up to four miscellaneous functions **M** at the end of an NC block or in a separate NC block. Once you confirm the entry of a miscellaneous function, the control continues with the dialog and you can define additional parameters, such as **M140 MB MAX**.

In the **Manual operation** application, use the **M** button to activate a miscellaneous function.

Further information: "Manual operation application", Page 202

Effects of the miscellaneous functions M

Miscellaneous functions **M** are in effect blockwise or modally. Miscellaneous functions take effect from their point of definition. Other functions or the end of the NC program reset modally effective 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.

If you program more than one miscellaneous function in an NC block, the execution sequence is as follows:

- Miscellaneous functions taking effect at the start of the block are executed before those taking effect at the end of the block.
- If more than one miscellaneous function takes effect at the start or end of the block, they are executed in the same sequence as programmed.

STOP function

The **STOP** function interrupts the program run or simulation (e.g., for tool inspection). You can also enter up to four miscellaneous functions **M** in a **STOP** block.

23.1.1 Programming the STOP function

To program the **STOP** function:

STOP

- ▶ Select **STOP**
- > The control creates a new NC block with the **STOP** function.

23.2 Overview of miscellaneous functions

Refer to your machine manual.

The machine manufacturer can influence the behavior of the miscellaneous functions described below.

M0 to **M30** are standardized miscellaneous functions.

This table shows at what point the miscellaneous functions take effect:

□ At the start of the block

■ At the end of the block

Function	Effect	Further information
M0 Stop program run and the spindle, switch coolant supply off	■	
M1 Optionally stop program run, optionally stop the spindle, optionally switch the coolant supply off Function depends on the machine manufacturer	■	
M2 Stop program run and the spindle, switch coolant supply off, return to beginning of the program, optionally reset the program information The functions depends on the setting by the machine manufacturer in the machine parameter **resetAt** (no. 100901)	■	
M3 Switch spindle on clockwise	□	
M4 Switch spindle on counterclockwise	□	
M5 Stop the spindle	■	
M8 Switch coolant supply on	□	
M9 Switch coolant supply off	■	
M13 Switch spindle on clockwise, switch coolant supply on	□	
M14 Switch spindle on counterclockwise, switch coolant supply on	□	
M30 Function is Identical to **M2**	■	

Function	Effect	Further information
M89 Free miscellaneous function **or** Call cycle modally Function depends on the machine manufacturer	□ ■	Page 481
M91 Traverse in the machine coordinate system **M-CS**	□	Page 1322
M92 Traverse in the **M92** coordinate system	□	Page 1323
M94 Reduce the display for rotary axes to under 360°	□	Page 1325
M97 Machine small contour steps	■	Page 1327
M98 Machine open contours completely	■	Page 1329
M99 Call a cycle once per block	■	Page 481
M101 Automatically insert a replacement tool	□	Page 1355
M102 Reset **M101**	■	
M103 Reduce feed rate for infeed movements	□	Page 1330
M107 Permit positive tool oversizes	□	Page 1357
M108 Check the radius of the replacement tool Reset **M107**	■	Page 1359
M109 Adapt feed rate for circular paths	□	Page 1331
M110 Reduce feed rate for inner radii	□	
M111 Reset **M109** and **M110**	■	
M116 Interpret feed rate for rotary axes as mm/min	□	Page 1333
M117 Reset **M116**	■	
M118 Activate handwheel superimpositioning	□	Page 1334
M120 Pre-calculate the radius-compensated contour (look ahead)	□	Page 1336

Function	Effect	Further information
M126 Shorter-path traverse of rotary axes	□	Page 1340
M127 Reset **M126**	■	
M128 Automatically compensate for tool inclination (TCPM)	□	Page 1341
M129 Reset **M128**	■	
M130 Traverse in the non-tilted input coordinate system **I-CS**	□	Page 1324
M136 Interpret feed rate as mm/rev	□	Page 1346
M137 Reset **M136**	■	
M138 Take rotary axes into account during machining operations	□	Page 1347
M140 Retract in the tool axis	□	Page 1348
M141 Suppress touch probe monitoring	□	Page 1360
M143 Rescind basic rotations	□	Page 1350
M144 Factor the tool offset into the calculations	□	Page 1350
M145 Reset **M144**	■	
M148 Automatically lift off upon an NC stop or a power failure	□	Page 1352
M149 Reset **M148**	■	
M197 Prevent rounding off of outside corners	■	Page 1353

23.3 Miscellaneous functions for coordinate entries

23.3.1 Traversing in the machine coordinate system M-CS with M91

Application

You can use **M91** to program machine-based positions, such as for moving to safe positions. The coordinates of positioning blocks with **M91** are effective in the machine coordinate system **M-CS**.

Further information: "Machine coordinate system M-CS", Page 1012

Description of function

Effect

M91 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 L Z+250 R0 FMAX M91	; Approach a safe position in the tool axis
13 L X-200 Y+200 R0 FMAX M91	; Approach a safe position in the plane
14 LBL 0	

Here **M91** is in a subprogram in which the control moves the tool to a safe position, by first moving in the tool axis and then in the plane.

Since the coordinates refer to the machine datum, the tool always moves to the same position. That way, regardless of the workpiece preset, the subprogram can be repeatedly called in the NC program, for example before tilting the rotary axes.

Without **M91** the control references the programmed coordinates to the workpiece preset.

Further information: "Presets in the machine", Page 210

> The coordinates for a safe position depend on the machine.
> The machine manufacturer defines the position of the machine datum.

Notes

- 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**. For the first position programmed with **M91**, the incremental coordinates are relative to the current tool position.
- The control considers any active tool radius compensation when positioning with **M91**.

  Further information: "Tool radius compensation", Page 1114
- The control uses the tool carrier reference point when positioning in the tool axis.

  Further information: "Presets in the machine", Page 210
- The following position displays refer to the machine coordinate system **M-CS** and show the values defined with **M91**:
  - **Nominal reference position (RFNOML)**
  - **Actual reference position (RFACTL)**

  Further information: "Position displays", Page 188
- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset for the simulation. In this constellation you can simulate traverse movements with **M91**.

  Further information: "Visualization options column", Page 1537
- In the machine parameter **refPosition** (no. 400403) the machine manufacturer defines the position of the machine datum.

23.3.2 Traversing in the M92 coordinate system with M92

Application

You can use **M92** to program machine-based positions, such as for moving to safe positions. The coordinates of positioning blocks with **M92** are relative to the **M92** datum and are effective in the **M92** coordinate system.

Further information: "Presets in the machine", Page 210

Description of function

Effect

M92 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 L Z+0 R0 FMAX M92	; Approach a safe position in the tool axis
13 L X+0 Y+0 R0 FMAX M92	; Approach a safe position in the plane
14 LBL 0	

Here **M92** is in a subprogram in which the tool moves to a safe position, by first moving in the tool axis and then in the plane.

Since the coordinates refer to the **M92** datum, the tool always moves to the same position. That way, regardless of the workpiece preset, the subprogram can be repeatedly called in the NC program, for example before tilting the rotary axes.

Without **M92** the control references the programmed coordinates to the workpiece preset.

Further information: "Presets in the machine", Page 210

The coordinates for a safe position depend on the machine.
The machine manufacturer defines the position of the **M92** datum.

Notes

- The control considers any active tool radius compensation when positioning with **M92**.

  Further information: "Tool radius compensation", Page 1114
- The control uses the tool carrier reference point when positioning in the tool axis.

  Further information: "Presets in the machine", Page 210
- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset for the simulation. In this constellation you can simulate traverse movements with **M92**.

  Further information: "Visualization options column", Page 1537
- In the optional machine parameter **distFromMachDatum** (no. 300501) the machine manufacturer defines the position of the **M92** datum.

23.3.3 Traversing in the non-tilted input coordinate system I-CS with M130

Application

Coordinates of a straight line entered with **M130** are effective in the non-tilted input coordinate system **I-CS** despite a tilted working plane, such as for retraction.

Description of function

Effect

M130 is in effect blockwise for straight lines without radius compensation and takes effect at the start of the block.

Further information: "Straight line L", Page 326

Application example

11 L Z+20 R0 FMAX M130	; Retract in the tool axis

With **M130**, the control references the coordinates in this NC block to the non-tilted input coordinate system **I-CS** despite a tilted working plane. That way the control retracts the tool perpendicular to the top edge of the workpiece.

Without **M130** the control references the coordinates of the straight line to the tilted **I-CS**.

Further information: "Input coordinate system I-CS", Page 1021

Notes

NOTICE

Danger of collision!

The miscellaneous function **M130** is effective only blockwise. The control executes the subsequent machining operations in the tilted working plane coordinate system **WPL-CS** again. Danger of collision during machining!

- Use the simulation to check the sequence and positions

If you combine **M130** with a cycle call, the control will interrupt machining with an error message.

Definition

Non-tilted input coordinate system I-CS
In a non-tilted input coordinate system **I-CS** the control ignores the tilting of the working plane, but does take into account the alignment of the workpiece's upper surface and all active transformations, such as a rotation.

23.4 Miscellaneous functions for path behavior

23.4.1 Reducing the display for rotary axes to under 360° with M94

Application

With **M94** the control reduces the display of the rotary axes to a range between 0° and 360°. Additionally, this limitation reduces the angle difference between the actual position and the new nominal position to less than 360°, which shortens traverse movements.

Related topics

- Values of the rotary axes in the position display

  Further information: "Positions workspace", Page 163

Description of function

Effect

M94 is in effect blockwise and takes effect at the start of the block.

Application example

11 L IC+420	; Move the C axis
12 L C+180 M94	; Reduce the display value of the C axis and move the axis

Before machining, the control shows the value 0° in the position display of the C axis.

In the first NC block the C axis moves incrementally by 420°, for example in order to cut an adhesive slot.

The second NC block first reduces the display of the C axis from 420° to 60°. Then the control positions the C axis to the nominal position of 180°. The angle difference is now 120°.

Without **M94** the angle difference would be 240°.

Input

If you define **M94**, the control continues the dialog and prompts you for the affected rotary axis. If you do not enter an axis, the control reduces the position display for all rotary axes.

21 L M94	; Reduce the display values of all rotary axes

21 L M94 C	; Reduce the display value of the C axis

Notes

- **M94** only affects rollover axes whose actual position display permits values above 360°.
- In the machine parameter **isModulo** (no. 300102) the machine manufacturer defines whether the modulo counting method is used for a rollover axis.
- In the optional machine parameter **shortestDistance** (no. 300401) the machine manufacturer defines whether the control by default positions the rotary axis on the shortest traverse path.
- In the optional machine parameter **startPosToModulo** (no. 300402) the machine manufacturer defines whether the control reduces the actual position display to a range between 0° and 360° before each positioning.
- If traverse limits or software limit switches are active for a rotary axis then **M94** has no effect on this rotary axis.

Definitions

Modulo axis

Modulo axes are axes whose encoder only returns values between 0° and 359.9999°. If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

Rollover axis

Rollover axes are rotary axes that can perform several or any number of revolutions. The machine manufacturer must configure a rollover axis as a modulo axis.

Modulo counting method

The position display of a rotary axis with the modulo counting method is between 0° and 359.9999°. If the value exceeds 359.9999°, the display starts over at 0°.

23.4.2 Machining small contour steps with M97

Application

With **M97** you can produce contour steps that are smaller than the tool radius. The control does not damage the contour and does not issue an error message.

> Instead of **M97**, HEIDENHAIN recommends using the more powerful function **M120** (option 21).
>
> After activating **M120** you can produce complete contours without error messages. **M120** also considers circular paths.

Related topics

- Pre-calculating a radius-compensated contour with **M120**

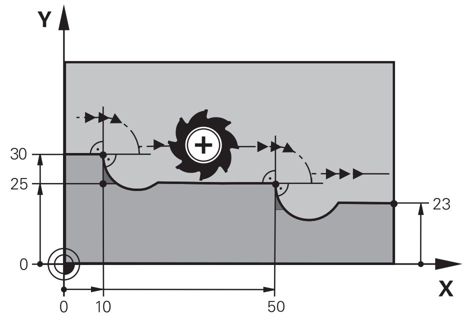
  Further information: "Pre-calculating a radius-compensated contour with M120", Page 1336

Description of function

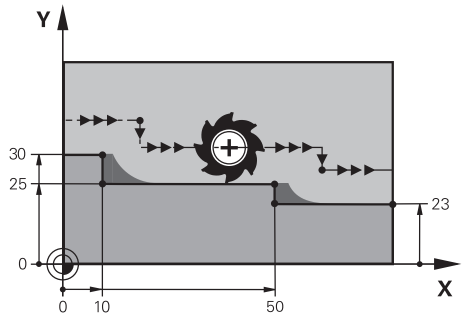
Effect

M97 is in effect blockwise and takes effect at the end of the block.

Application example



Contour step without **M97**



Contour step with **M97**

11 TOOL CALL 8 Z S5000	; Insert the tool with diameter 16
*** - ...**	
21 L X+0 Y+30 RL	
22 L X+10 M97	; Machine the contour step using the path intersection
23 L Y+25	
24 L X+50 M97	; Machine the contour step using the path intersection
25 L Y+23	
26 L X+100	

For radius-compensated contour steps, the control uses **M97** to determine a path intersection that is in the extension of the tool path. The control extends the tool path each time by the tool radius. This means that the smaller the counter step is and the larger the tool radius, the greater the contour extension is. The control moves the tool beyond the path intersection and thus avoids damage to the contour.

Without **M97** the tool would move on a transitional arc around the outside corners and damage the contour. At such locations the control interrupts machining with the **Tool radius too large** error message.

Notes

- Program **M97** only for outside corners.
- For further machining operations, please note that shifting the contour corner results in more residual material. You may then need to rework the contour step with a smaller tool.

23.4.3 Machining open contour corners with M98

Application

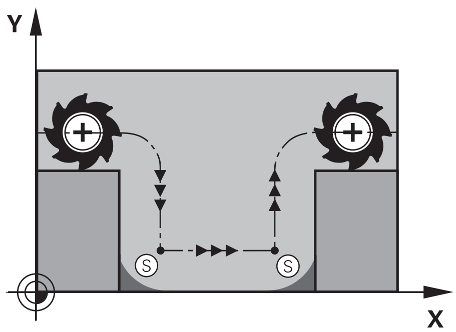
If the tool performs a machining operation on a radius-compensated contour, then residual material remains at the inside corners. With **M98** the control extends the tool path by the tool radius so that the tool completely machines an open contour and removes all residual material.

Description of function

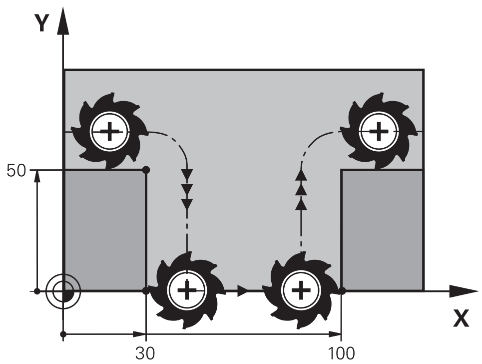
Effect

M98 is in effect blockwise and takes effect at the end of the block.

Application example



Open contour without **M98**



Open contour with **M98**

11 L X+0 Y+50 RL F1000	
12 L X+30	
13 L Y+0 M98	; Completely machine an open contour corner
14 L X+100	; The control maintains the position of the Y axis with **M98**
15 L Y+50	

The control moves the tool along the contour with radius compensation. With **M98** the control calculates the contour ahead of time and determines a new path intersection in the extension of tool path. The control moves the tool beyond this path intersection and completely machines the open contour.

In the next NC block the control maintains the position of the Y axis.

Without **M98** the control uses the programmed coordinates as limitation for the radius-compensated contour. The control calculates the path intersection so that the contour is not damaged and residual material remains.

23.4.4 Reducing the feed rate for infeed movements with M103

Application

With **M103** the control performs infeed movements at a lower feed rate, for example when plunging. You use a percent factor to define the feed-rate value.

Description of function

Effect

M103 is in effect for straight lines in the tool axis at the start of the block.

In order to reset **M103**, program **M103** without a defined factor.

Application example

11 L X+20 Y+20 F1000	; Move in the working plane
12 L Z-2.5 M103 F20	; Activate feed rate reduction and move at reduced feed rate
12 L X+30 Z-5	; Move at reduced feed rate

In the first NC block the control positions the tool in the working plane.

In NC block **12** the control activates **M103** with the percent factor 20 and then performs the infeed movement in the Z axis at a reduced feed rate of 200 mm/min.

Next, in NC block **13**, the control performs an infeed movement in the X and Z axes at a reduced feed rate of 825 mm/min. This higher feed rate results from the control moving the tool in the plane in addition to the infeed movement. The control calculates a cutting value between the feed rate in the plane and the infeed rate.

Without **M103** the infeed movement is performed at the programmed feed rate.

Input

If you define **M103**, the control continues the dialog and prompts you for the factor **F**.

Notes

- The infeed rate F_Z is calculated from the last programmed feed rate F_{Prog} and the percent factor **F**.

  $F_Z = F_{Prog} \times F$

- **M103** is also in effect with an active tilted working plane coordinate system **WPL-CS**. The feed rate reduction is then active during infeed movements in the virtual tool axis **VT**.

23.4.5 Adapting the feed rate for circular paths with M109

Application

With **M109** the control maintains a constant feed rate at the cutting edge for internal and external machining on circular paths, for example to produce a uniform milled surface during finishing.

Description of function

Effect

M109 takes effect at the start of the block.

In order to reset **M109**, program **M111**.

Application example

11 L X+5 Y+25 RL F1000	; Approach first contour point at programmed feed rate
12 CR X+45 Y+25 R+20 DR- M109	; Activate feed rate adaptation, then perform the operation on the circular path at the increased feed rate

In the first NC block the control moves the tool at the programmed feed rate, which refers to the tool center-point path.

In NC block **12** the control activates **M109** and maintains a constant feed rate at the tool cutting edge when machining on circular paths. At the beginning of each block the control calculates the feed rate at the tool cutting edge for the respective NC block and adapts the programmed feed rate depending on the contour radius and tool radius. This means that the programmed feed rate is increased for external operations and reduced for internal operations.

The tool then cuts the external contour at an increased feed rate.

Without **M109** the tool cuts along the circular path at the programmed feed rate.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

If the **M109** function is active, the control might significantly increase the feed rate when machining very small outside corners (acute angles). There is a risk of tool breakage or workpiece damage during machining.

- Do not use **M109** for machining very small outside corners (acute angles)

If you define **M109** before calling a machining cycle with a number greater than **200**, the adjusted feed rate is also active for circular paths within these machining cycles.

23.4.6 Reducing the feed rate for internal radii with M110

Application

With **M110** the control maintains a constant feed rate at the cutting edge only for internal radii, as opposed to **M109**. This results in consistent cutting conditions affecting the tool, which is important, for example, in heavy-duty machining.

Description of function

Effect

M110 takes effect at the start of the block.

In order to reset **M110**, program **M111**.

Application example

11 L X+5 Y+25 RL F1000	; Approach first contour point at programmed feed rate
12 CR X+45 Y+25 R+20 DR+ M110	; Activate feed rate reduction, then perform the operation on the circular path at the reduced feed rate

In the first NC block the control moves the tool at the programmed feed rate, which refers to the tool center-point path.

In NC block **12** the control activates **M110** and maintains a constant feed rate at the tool cutting edge when machining on internal radii. At the beginning of each block the control calculates the feed rate at the tool cutting edge for the respective NC block and adapts the programmed feed rate depending on the contour radius and tool radius.

The tool then cuts the internal radius at a reduced feed rate.

Without **M110** the tool cuts along the internal radius at the programmed feed rate.

Note

If you define **M110** before calling a machining cycle with a number greater than **200**, the adjusted feed rate is also active for circular paths within these machining cycles.

23.4.7 Interpreting the feed rate for rotary axes as mm/min with M116 (option 8)

Application

With **M116** the control interprets the feed rate for rotary axes as millimeters per minute.

Requirements

- Machine with rotary axes
- Kinematics description

> Refer to your machine manual.
> The machine manufacturer creates the kinematics description of the machine.

- Software option 8: Advanced Functions (set 1)

Description of function

Effect

M116 is active only in the working plane and takes effect at the start of the block.

In order to reset **M116**, program **M117**.

Application example

11 L IC+30 F500 M116	; Move in the C axis in mm/min

With **M116** the control interprets the programmed feed rate of the C axis as mm/min, such as for cylinder surface machining.

In this case, the control calculates the feed for the block at the start of each NC block, taking the distance from the tool center point to the center of the rotary axis into account.

The feed rate does not change while the control is executing the NC block. This also applies for when the tool is moving towards the center of a rotary axis.

Without **M116** the control interprets the feed rate programmed for a rotary axis as degrees per minute.

Notes

- You can program **M116** for head and table rotary axes.
- The **M116** function also has an effect if the **Tilt working plane** function is active.

  Further information: "Tilting the working plane (option 8)", Page 1053
- It is not possible to combine **M116** with **M128** or **FUNCTION TCPM** (option 9). If you want to activate **M116** for an axis while **M128** or **FUNCTION TCPM** is active, then you must use **M138** to exclude this axis before machining.

  Further information: "Taking rotary axes into account during machining operations with M138", Page 1347
- Without **M128** or **FUNCTION TCPM** (option 9), **M116** can take effect for multiple rotary axes at the same time.

23.4.8 Activating handwheel superimpositioning with M118

Application

With **M118** the control activates handwheel superimpositioning. You can then perform manual corrections by handwheel during program run.

Related topics

- Handwheel superimpositioning with global program settings GPS (option 44)
  Further information: "Function Handwheel superimp.", Page 1225

Requirements

- Handwheel
- Software option 21: Advanced Functions (set 3)

Description of function

Effect

M118 takes effect at the start of the block.

In order to reset **M118**, program **M118** without entering any axes.

> Canceling a program also resets handwheel superimpositioning.

Application example

11 L Z+0 R0 F500	; Move in the tool axis
12 L X+200 R0 F250 M118 Z1	; Move in the working plane with active handwheel superimpositioning of no more than ±1 mm in the Z axis

In the first NC block the control positions the tool in the tool axis.

In NC block **12** the control activates handwheel superimpositioning at the start of the block with a maximum traverse range of ±1 mm in the Z axis.

Then the control performs the traverse movement in the working plane. During this traverse movement you can use the handwheel for continuous motion of the tool in the Z axis by up to ±1 mm. This way you can, for example, rework a workpiece that has been reclamped but that cannot be probed due to its free-form surface.

Input

If you define **M118**, the control continues the dialog and prompts you for the axes and the maximum permissible superimpositioning value. For linear axes you define the value in millimeters and for rotary axes in degrees.

21 L X+0 Y+38.5 RL F125 M118 X1 Y1	; Move in the working plane with active handwheel superimpositioning of no more than ±1 mm in the X and Y axes

Notes

> Refer to your machine manual.
> Your machine manufacturer must have prepared the control for this function.

- By default **M118** is in effect in the machine coordinate system **M-CS**.

  If you activate the **handwheel superimpositioning** switch in the **GPS** (option 44) workspace, handwheel superimpositioning is active in the last selected coordinate system.

  Further information: "Global Program Settings (GPS, option 44)", Page 1217
- On the **POS HR** tab of the **Status** workspace the control shows the active coordinate system in which handwheel superimpositioning is in effect, as well as the maximum possible traverse values of the respective axes.

  Further information: "POS HR tab", Page 180
- The handwheel superimpositioning function with **M118** in combination with dynamic collision monitoring (DCM, option 40) is possible only at a standstill.

  To be able to use **M118** without restrictions, you have to deactivate **DCM** (option 40) or activate a kinematics model without collision objects.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
- Handwheel superimpositioning is also effective in the **MDI** application.

  Further information: "The MDI Application ", Page 1933
- If you want to use **M118** with clamped axes, you must unclamp them first.

Notes in conjunction with the virtual tool axis VT (option 44)

> Refer to your machine manual.
> Your machine manufacturer must have prepared the control for this function.

- On machines with head rotation axes, you can choose for inclined machining whether superimpositioning should be in effect in the Z axis or along the virtual tool axis **VT**.
- In the machine parameter **selectAxes** (no. 126203) the machine manufacturer defines the assignment of axis keys on the handwheel.

  When using an HR 5xx handwheel, you can assign the virtual axis to the orange **VI** axis key, if desired.

23.4.9 Pre-calculating a radius-compensated contour with M120

Application

With **M120** the control pre-calculates a radius-compensated contour. This way the control can produce contours that are smaller than the tool radius without damaging the contour or issuing an error message.

Requirement

- Software option 21: Advanced Functions (set 3)

Description of function

Effect

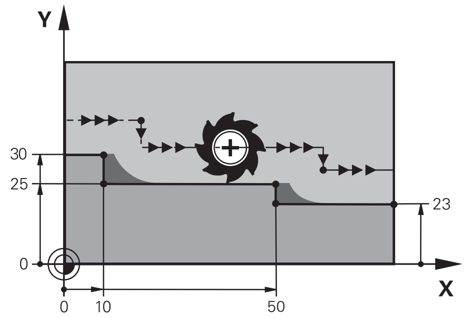
M120 takes effect at the start of the block and remains active beyond the milling cycles.

The following functions reset **M120**:

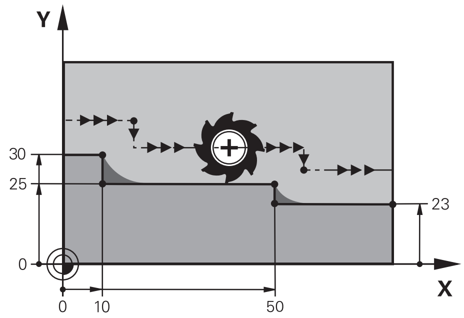
- Radius compensation **R0**
- **M120 LA0**
- **M120** without **LA**
- **PGM CALL**
- **PLANE** functions (option 8)
- Cycle **19 WORKING PLANE**

> You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

Application example



Contour step with **M97**



Contour step with **M120**

11 TOOL CALL 8 Z S5000	; Insert the tool with diameter 16
*** - ...**	
21 L X+0 Y+30 RL M120 LA2	; Activate contour pre-calculation and move in the working plane
22 L X+10	
23 L Y+25	
24 L X+50	
25 L Y+23	
26 L X+100	

With **M120 LA2** in NC block **21**, the control checks the radius-compensated contour for undercuts. In this example the control calculates the tool path starting from the current NC block for two NC blocks at a time. Then the control uses radius compensation while positioning the tool to the first contour point.

When machining the contour, the control extends the tool path in each case so that the tool does not damage the contour.

Without **M120** the tool would move on a transitional arc around the outside corners and damage the contour. At such locations the control interrupts machining with the **Tool radius too large** error message.

Input

If you define **M120**, the control continues the dialog and prompts you for the number of **LA** NC blocks to be calculated in advance (up to 99).

Notes

NOTICE

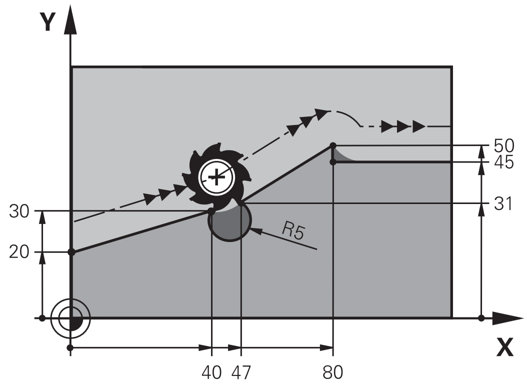
Danger of collision!

Define as low a number as possible of **LA** NC blocks to be pre-calculated. If the value defined is too large, the control might overlook parts of the contour!

- Use the Simulation mode to test the NC program before execution
- Verify the NC program by slowly executing it block by block

- For further machining operations, please note that residual material remains in the contour corners. You may then need to rework the contour step with a smaller tool.
- If you always program **M120** in the same NC block as the radius compensation you can achieve consistent and clearly structured programs.
- If you run the following functions while **M120** is active, then the control cancels program run and issues an error message:
  - Cycle **32 TOLERANCE**
  - **M128** (option 9)
  - **FUNCTION TCPM** (option 9)
  - Mid-program startup

Example



0 BEGIN PGM "M120" MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-10	
2 BLK FORM 0.2 X+110 Y+80 Z+0	; Workpiece blank definition
3 TOOL CALL 6 Z S1000 F1000	; Insert the tool with diameter 12
4 L X-5 Y+26 R0 FMAX M3	; Move in the working plane
5 L Z-5 R0 FMAX	; Infeed in the tool axis
6 L X+0 Y+20 RL F AUTO M120 LA5	; Activate contour pre-calculation and move to the first contour point
7 L X+40 Y+30	
8 CR X+47 Y+31 R-5 DR+	
9 L X+80 Y+50	
10 L X+80 Y+45	
11 L X+110 Y+45	; Move to the last contour point
12 L Z+100 R0 FMAX M120	; Retract the tool and reset **M120**
13 M30	; End of program
14 END PGM "M120" MM	

Definition

Abbreviation	Definition
LA (look ahead)	Number of look-ahead blocks

23.4.10 Shorter-path traversing of rotary axes with M126

Application

With **M126** the control moves a rotary axis on the shortest path of traverse to the programmed coordinates. This function affects only rotary axes whose position display is reduced to a value of less than 360°.

Description of function

Effect

M126 takes effect at the start of the block.

In order to reset **M126**, program **M127**.

Application example

11 L C+350	; Move in the C axis
12 L C+10 M126	; Shortest-path traverse in the C axis

In the first NC block the control positions the C axis to 350°.

In the second NC block the control activates **M126** and then positions the C axis with shortest-path traverse to 10°. The control uses the shortest traverse path and moves the C axis in the positive direction of rotation, beyond 360°. The traverse path is 20°.

Without **M126** the control does not move the rotary axis beyond 360°. The traverse path is then 340° in the negative direction of rotation.

Notes

- **M126** is not in effect with incremental traverse movements.
- The effect of **M126** depends on the configuration of the rotary axis.
- **M126** has an effect only on modulo axes.

  In the machine parameter **isModulo** (no. 300102) the machine manufacturer defines whether a rotary axis is a modulo axis.
- In the optional machine parameter **shortestDistance** (no. 300401) the machine manufacturer defines whether the control by default positions the rotary axis on the shortest traverse path.
- In the optional machine parameter **startPosToModulo** (no. 300402) the machine manufacturer defines whether the control reduces the actual position display to a range between 0° and 360° before each positioning.

Definitions

Modulo axis

Modulo axes are axes whose encoder only returns values between 0° and 359.9999°. If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

Rollover axis

Rollover axes are rotary axes that can perform several or any number of revolutions. The machine manufacturer must configure a rollover axis as a modulo axis.

Modulo counting method

The position display of a rotary axis with the modulo counting method is between 0° and 359.9999°. If the value exceeds 359.9999°, the display starts over at 0°.

23.4.11 Automatically compensating for tool inclination with M128 (option 9)

Application

If the position of a controlled rotary axis changes in the NC program, then the control uses **M128** during the tilting procedure to automatically compensate for the tool inclination with a compensating movement of the linear axes. That way the position of the tool tip relative to the workpiece surface remains unchanged (TCPM).

> Instead of **M128**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM**.

Related topics

- Compensating for tool offset with **FUNCTION TCPM**

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

Requirement

- Machine with rotary axes
- Kinematics description

  > Refer to your machine manual.
  >
  > The machine manufacturer creates the kinematics description of the machine.

- Software option 9: Advanced Functions (set 2)

Description of function

Effect

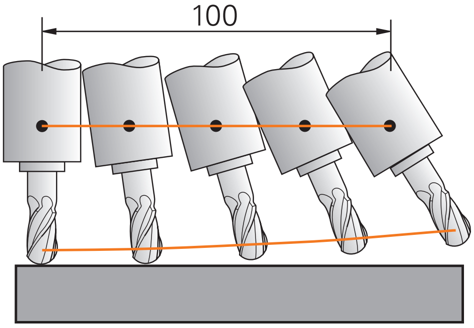
M128 takes effect at the start of the block.

You can reset **M128** with the following functions:

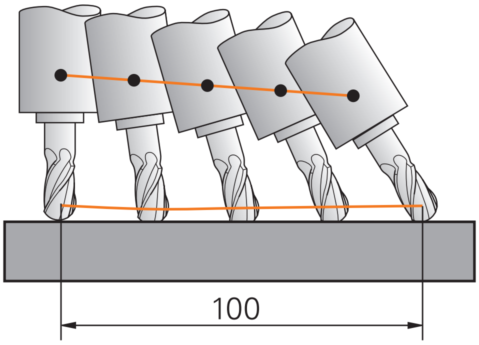
- **M129**
- **FUNCTION RESET TCPM**
- In the **Program Run** operating mode, select a different NC program

> **M128** is also in effect in the **Manual** operating mode and remains active even after a change in the operating mode.

Application example



Behavior without **M128**



Behavior with **M128**

11 L X+100 B-30 F800 M128 F1000	; Move with automatic compensation of the motion in the rotary axis

In this NC block the control activates **M128** with the feed rate for the compensating movement. The control then simultaneously moves the tool in the X axis and in the B axis.

In order to keep the position of the tool tip constant relative to the workpiece while inclining the rotary axis, the control uses the linear axes to perform a continuous compensating movement. In this example the control performs the compensating movement in the Z axis.

Without **M128** an offset of the tool tip relative to the nominal position results as soon as the inclination angle of the tool changes. The control does not compensate for this offset. If you do not take this deviation into account in the NC program, the machining operation will not be performed correctly or a collision will occur.

Input

If you define **M128**, the control continues the dialog and prompts you for the feed rate **F**. The defined value limits the feed rate during the compensating movement.

Inclined machining with open-loop rotary axes

With open-loop rotary axes, also known as counter axes, you can also perform inclined machining in combination with **M128**.

For inclined machining operations with open-loop rotary axes, proceed as follows:

- Before activating **M128**, position the rotary axes manually
- Activate **M128**
- > The control reads the actual values of all existing rotary axes, calculates from this the new position of the tool location point, and updates the position display.

  Further information: "Presets on the tool", Page 271
- > The control performs the necessary compensating movement with the next traverse movement.
- Execute the machining operation
- Reset **M128** at the program end with **M129**
- Return the rotary axes to their initial position

> As long as **M128** is active, the control monitors the actual positions of the open-loop rotary axes. If the actual position deviates from the value that is defined by the machine manufacturer, then the control issues an error message and interrupts program run.

Notes

NOTICE
Danger of collision! Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation. ▶ Make sure to retract the tool before changing the position of the rotary axis

NOTICE
Danger of collision! For peripheral milling, if you define the tool inclination using **LN** straight lines with tool orientation **TX**, **TY**, and **TZ**, the control autonomously calculates the required positions of the rotary axes. This can result in unexpected movements. ▶ Use the Simulation mode to test the NC program before execution ▶ Verify the NC program by slowly executing it block by block

Further information: "3D tool compensation during peripheral milling (option 9)", Page 1137

Further information: "Output with vectors", Page 1303

- The feed rate for the compensating movement remains in effect until you program a new feed rate or rescind **M128**.
- If **M128** is active, the control shows the **TCPM** symbol in the **Positions** workspace.

  Further information: "Positions workspace", Page 163
- You define the inclination angle of the tool by entering the axis positions of the rotary axes directly. This way the values refer to the machine coordinate system **M-CS**. For machines with head rotation axes the tool coordinate system **T-CS** changes. For machines with table rotary axes the workpiece coordinate system **W-CS** changes.

  Further information: "Reference systems", Page 1010
- If you run the following functions while **M128** is active, then the control cancels program run and issues an error message:
  - Cutting edge radius compensation **RR/RL** in turning operations (option 50)
  - **M91**
  - **M92**
  - **M144**
  - Calling a tool with **TOOL CALL**
  - Dynamic Collision Monitoring (DCM, option 40) and at the same time **M118**

Notes about machine parameters

- In the optional machine parameter **maxCompFeed** (no. 201303), the machine manufacturer defines the maximum speed of compensating movements.
- In the optional machine parameter **maxAngleTolerance** (no. 205303), the machine manufacturer defines the maximum angle tolerance.
- In the optional machine parameter **maxLinearTolerance** (no. 205305), the machine manufacturer defines the maximum linear axis tolerance.
- In the optional machine parameter **manualOversize** (no. 205304), the machine manufacturer defines a manual oversize for all collision objects.
- The machine manufacturer uses the optional machine parameter **preset-ToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. For **FUNCTION TCPM** and **M128**, the machine parameter applies only to the rotary axis that rotates about the tool axis (in most cases **C_OFFS**).

  Further information: "Basic transformation and offset", Page 2039

  - If the machine parameter axis has not been defined or has been set to **TRUE**, the offset can be used to compensate a misalignment of the workpiece in the plane. The offset affects the orientation of the workpiece coordinate system **W-CS**.

    Further information: "Workpiece coordinate system W-CS", Page 1016
  - If the machine parameter axis has been defined with **FALSE**, the offset cannot be used to compensate a misalignment of the workpiece in the plane. The control will not take the offset into account when executing the commands.

Notes on tools

If you incline a tool while machining a contour, you must use a ball-nose cutter; otherwise the tool can damage the contour.

In order to avoid damaging a contour while machining it with a ball-nose cutter, note the following:

- With **M128** the control equates the tool rotation point with the tool location point. If the tool rotation point is at the tool tip, the tool will damage the contour if the tool is inclined. Therefore the tool location point must be at the tool center point.

  Further information: "Presets on the tool", Page 271
- In order for the control to display the tool correctly in the simulation, you must define its actual length in the column **L** of the tool management.

  When calling the tool in the NC program, define the sphere radius as a negative delta value in **DL** and thus shift the tool location point to the tool center point.

  Further information: "Tool length compensation", Page 1113

  For Dynamic Collision Monitoring (DCM, option 40) it is also important that you define the tool's actual length in the tool management.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1164
- If the tool location point is at the tool center point you must modify the coordinates of the tool axis in the NC program by the value of the sphere radius.

In **FUNCTION TCPM** you can choose the tool location point and the tool rotation point separately from each other.

Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

Definition

Abbreviation	Definition
TCPM (tool center point management)	Maintain the position of the tool location point **Further information:** "Presets on the tool", Page 271

23.4.12 Interpreting the feed rate as mm/rev with M136

Application

With **M136** the control interprets the feed rate as millimeters per revolution. The feed rate depends on the spindle speed, for example in conjunction with the turning mode (option 50).

Further information: "Switching the operating mode with FUNCTION MODE", Page 234

Description of function

Effect

M136 takes effect at the start of the block.

In order to reset **M136**, program **M137**.

Application example

11 LBL "TURN"	
12 FUNCTION MODE TURN	; Activate turning mode
13 M136	; Switch interpretation of the feed rate to mm/rev
14 LBL 0	

M136 is here located in a subprogram in which the control activates the turning mode (option 50).

With **M136** the control interprets the feed rate as millimeters per spindle revolution, which is necessary for the turning mode. The feed rate per revolution refers to the rotational speed of the workpiece spindle. The control thus moves the tool at the programmed feed rate for every rotation of the workpiece spindle.

Without **M136** the control interprets the feed rate as millimeters per minute.

Notes

- 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.
- **M136** is not possible in combination with an oriented spindle stop. The control cannot calculate the feed rate because the spindle does not rotate during an oriented spindle stop, such as when tapping.

23.4.13 Taking rotary axes into account during machining operations with M138

Application

With **M138** you define which rotary axes the control takes into account during the calculation and positioning of spatial angles. The control excludes any axes that were not defined. That way you can reduce the number of tilting possibilities and thus avoid error messages, for example on machines with three rotary axes.

M138 is in effect in combination with the following functions:

- **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1341
- **FUNCTION TCPM** (option 9)

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
- **PLANE** functions (option 8)

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 1054
- Cycle **19 WORKING PLANE** (option 8)

Description of function

Effect

M138 takes effect at the start of the block.

In order to reset **M138**, program **M138** without entering any rotary axes.

Application example

11 L Z+100 R0 FMAX M138 A C	; Define that axes **A** and **C** should be taken into account
12 PLANE SPATIAL SPA+0 SPB+90 SPC+0 MOVE FMAX	; Tilt spatial angle **SPB** by 90°

On a six-axis machine with **A**, **B**, and **C** rotary axes you must exclude one rotary axis for spatial angle operations; otherwise too many combinations are possible.

With **M138 A C** the control calculates the axis position when tilting with spatial angles only in the **A** and **C** axes. The B axis is excluded. Therefore, in NC block **12** the control positions the spatial angle **SPB+90** with the **A** and **C** axes.

Without **M138** there are too many possibilities for tilting. The control interrupts the machining process and issues an error message.

Input

If you define **M138**, the control continues the dialog and prompts you for the rotary axes to be taken into account.

11 L Z+100 R0 FMAX M138 C	; Define that the C axis should be taken into account

Notes

- With **M138** the control excludes the rotary axes only during the calculation and positioning of spatial angles. A rotary axis that has been excluded with **M138** can nevertheless be moved in a positioning block. Please note that in this case the control does not execute any compensations.
- In the optional machine parameter **parAxComp** (no. 300205) the machine manufacturer defines whether the control includes the position of the excluded axis when calculating the kinematics.

23.4.14 Retracting in the tool axis with M140

Application

With **M140** the control retracts the tool in the tool axis.

Description of function

Effect

M140 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 M140 MB MAX	; Retract by the maximum distance in the tool axis
13 L X+350 Y+400 R0 FMAX M91	; Approach a safe position in the working plane
14 LBL 0	

Here **M140** is in a subprogram in which the control moves the tool to a safe position.

With **M140 MB MAX** the control retracts the tool by the maximum distance in the positive direction in the tool axis. The control stops the tool before reaching a limit switch or a collision object.

In the next NC block the control moves the tool to a safe position in the working plane.

Without **M140** the control does not execute a retraction.

Input

If you define **M140**, the control continues the dialog and prompts you for the retraction distance **MB**. You can program the retraction distance as a positive or negative incremental value. With **MB MAX** the control retracts the tool in the positive direction in the tool axis before reaching a limit switch or a collision object.

After **MB** you can define a feed rate for the retraction movement. If you do not define a feed rate, the control retracts the tool at rapid traverse.

21 L Y+38.5 F125 M140 MB+50 F750	; Retract tool at feed rate of 750 mm/min by 50 mm in the positive direction of the tool axis

21 L Y+38.5 F125 M140 MB MAX	; Retract tool at rapid traverse by the maximum distance in the positive direction in the tool axis

Notes

NOTICE
Danger of collision! The machine manufacturer has various options for configuring the Dynamic Collision Monitoring (DCM, option 40) function. Depending on the machine, the control can continue with the NC program without an error message despite the detected collision. The control stops the tool at the last position without a collision and continues the NC program from this position. This configuration of DCM results in movements that are not defined in the program. **This behavior occurs no matter whether collision monitoring is active or inactive.** There is a danger of collision during these movements! ▶ Refer to your machine manual. ▶ Check the behavior at the machine.

NOTICE
Danger of collision! If you use **M118** to modify the position of a rotary axis with the handwheel and then execute **M140**, the control ignores the superimposed values during the retraction movement. This results in unwanted and unpredictable movements, especially when using machines with head rotation axes. There is a danger of collision during these retraction movements! ▶ Do not combine **M118** with **M140** when using machines with head rotation axes.

- **M140** is also in effect with a tilted working plane. For machines with head rotation axes the control moves the tool in the tool coordinate system **T-CS**.
  Further information: "Tool coordinate system T-CS", Page 1022
- With **M140 MB MAX** the control retracts the tool only in the positive direction in the tool axis.
- If you define a negative value for **MB**, the control retracts the tool in the negative direction in the tool axis.
- The control gleans the necessary information about the tool axis for **M140** from the tool call.
- In the optional machine parameter **moveBack** (no. 200903) the machine manufacturer defines the distance to a limit switch or a collision object upon a maximum retraction with **MB MAX**.

Definition

Abbreviation	Definition
MB (move back)	Tool axis retraction

23.4.15 Rescinding basic rotations with M143

Application

With **M143** the control resets a basic rotation as well as a 3D basic rotation, for example after machining a workpiece that needed alignment.

Description of function

Effect

M143 is in effect blockwise and takes effect at the start of the block.

Application example

11 M143	; Reset the basic rotation

In this NC block the control resets a basic rotation that had been defined in the NC program. In the active row of the preset table the control overwrites the values of the columns **SPA**, **SPB**, and **SPC** with the value **0**.

Without **M143** the basic rotation remains in effect until you manually reset the basic rotation or overwrite it with a new value.

Further information: "Preset management", Page 1025

Note

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

Further information: "Block scan for mid-program startup", Page 1965

23.4.16 Factoring the tool offset into the calculations with M144 (option 9)

Application

The control uses **M144** in subsequent traverse movements to compensate for tool offsets that result from inclined rotary axes.

> Instead of **M144**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM** (option 9).

Related topics

- Compensating for tool offset with **FUNCTION TCPM**

  Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104

Requirement

- Software option 9: Advanced Functions (set 2)

Description of function

Effect

M144 takes effect at the start of the block.

In order to reset **M144**, program **M145**.

Application example

11 M144	; Activate tool compensation
12 L A-40 F500	; Position the A axis
13 L X+0 Y+0 R0 FMAX	; Position the **X** and **Y** axes

With **M144** the control takes the position of the rotary axes into account in the subsequent positioning blocks.

In NC block **12** the control positions the rotary axis **A**, resulting in an offset between the tool tip and the workpiece. The control compensates for this offset mathematically.

In the next NC block the control positions the **X** and **Y** axes. When **M144** is active, the control compensates for the position of the rotary axis **A** during this movement.

Without **M144** the control does not take the offset into account, and the machining operation is performed with this offset.

Notes

Refer to your machine manual.

When working with angle heads, keep in mind that the machine geometry is defined by the machine manufacturer in a kinematics description.
If you use an angle head during machining, then you must select the correct kinematics description.

- You can use **M91** and **M92** for positioning even when **M144** is active.

  Further information: "Miscellaneous functions for coordinate entries", Page 1322
- The functions **M128** and **FUNCTION TCPM** are not permitted when **M144** is active. The control will issue an error message if you try to active these functions.
- **M144** does not work in connection with **PLANE** functions. If both functions are active, then the **PLANE** function is in effect.

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 1054

  With **M144** the control moves according to the workpiece coordinate system **W-CS**.

  If you activate **PLANE** functions, the control moves according to the working plane coordinate system **WPL-CS**.

  Further information: "Reference systems", Page 1010

Notes In conjunction with the turning operation (option 50)

- If the inclined axis is a tilting table, the control orients the tool coordinate system **W-CS**.

  If the inclined axis is a swivel head, the control does not orient the **W-CS**.
- After inclining the rotary axis, you may have to again pre-position the turning tool in the Y coordinate and orient the position of the tool tip with Cycle **800 ADJUST XZ SYSTEM**.

  Further information: "Cycle 800 ADJUST XZ SYSTEM ", Page 748

23.4.17 Automatically lifting off upon an NC stop or a power failure with M148

Application

With **M148** the control automatically retracts the tool from the workpiece in the following situations:

- Manually triggered NC stop
- NC stop triggered by the software, for example if an error has occurred in the drive system
- Power interruption

> Instead of **M148**, HEIDENHAIN recommends using the more powerful function **FUNCTION LIFTOFF**.

Related topics

- Automatic retraction with **FUNCTION LIFTOFF**
  Further information: "Automatic tool liftoff with FUNCTION LIFTOFF", Page 1191

Requirement

- **LIFTOFF** column in the tool management
  You must define the value **Y** in the **LIFTOFF** column of the tool management.
  Further information: "Tool management ", Page 297

Description of function

Effect

M148 takes effect at the start of the block.

You can reset **M148** with the following functions:

- **M149**
- **FUNCTION LIFTOFF RESET**

Application example

11 M148	; Activate automatic retraction

This NC block activates **M148**. If an NC stop is triggered during machining, the tool is retracted by up to 2 mm in the positive direction in the tool axis. This avoids possible damage due to the tool or workpiece.

Without **M148** the axes come to a stop upon an NC stop, meaning that the tool remains at the workpiece, which might result in surfaces blemishes on the workpiece.

Notes

- When lifting the tool off with **M148**, the control will not necessarily lift it off in the tool axis direction.

  The control uses the **M149** function to deactivate the **FUNCTION LIFTOFF** function without resetting the lift-off direction. If you program **M148**, the control will activate the automatic lift-off of the tool in the lift-off direction defined by the **FUNCTION LIFTOFF** function.
- Please note that for some tools, such as side milling cutters, automatic retraction does not make sense.
- In machine parameter **on** (no. 201401), the machine manufacturer defines whether automatic lift-off is active.
- In machine parameter **distance** (no. 201402), the machine manufacturer defines the maximum lift-off height.
- In the machine parameter **feed** (no. 201405), the machine manufacturer defines the speed of lift-off movement.

23.4.18 Preventing rounding off of outside corners with M197

Application

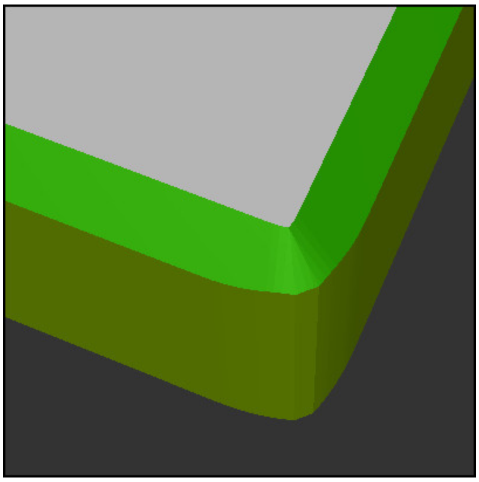
With **M197** the control extends a radius-compensated contour at the corner tangentially and inserts a smaller transition arc. That way you prevent the tool from rounding off the outside corner.

Description of function

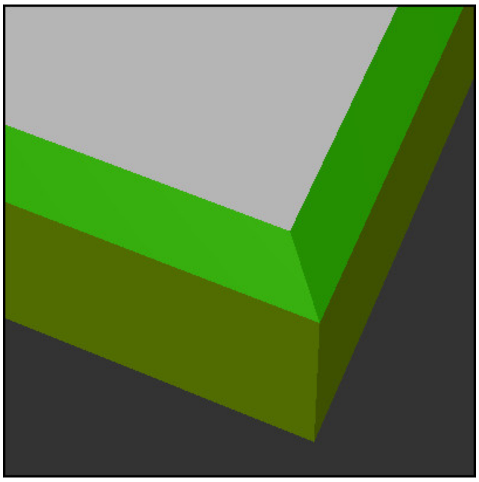
Effect

M197 is in effect blockwise and only for radius-compensated outside corners.

Application example



Contour without **M197**



Contour with **M197**

*** - ...**	; Approach the contour
11 X+60 Y+10 M197 DL5	; Machine the first contour with a sharp edge
12 X+10 Y+60 M197 DL5	; Machine the second contour with a sharp edge
*** - ...**	; Machine the remaining contour

With **M197 DL5** the control extends the contour at the corner tangentially by up to 5 mm. In this example, the 5 mm exactly correspond to the tool radius, resulting in an outside corner with a sharp edge. The control uses the smaller transitional arc to nevertheless move along the traverse path gently.

Without **M197** and with active radius compensation the control inserts a tangential transitional arc at an outside corner, which leads to rounding off of the outside corner.

Input

If you define **M197**, the control continues the dialog and prompts you for the tangential extension **DL**. **DL** is the maximum length by which the control extends the outside corner.

Note

In order to produce corners with sharp edges, define the parameter **DL** with the same size as the tool radius. The smaller the value you enter for **DL**, the more the corner will be rounded off.

Definition

Abbreviation	Definition
DL	Maximum tangential extension

23.5 Miscellaneous functions for tools

23.5.1 Automatically inserting a replacement tool with M101

Application

With **M101** the control automatically inserts a replacement tool after a specified tool life has expired. The control then continues the machining operation with the replacement tool.

Requirements

- **RT** column in the tool management

  The number of the replacement tool must have been defined in the **RT** column.
- **TIME2** column in the tool management

  In the **TIME2** column you define the tool life after which the control inserts the replacement tool.

Further information: "Tool management ", Page 297

Use only tools with an identical radius as replacement tools. The control does not automatically check the radius of the tool.

If you want the control to check the radius, program **M108** after the tool change.

Further information: "Checking the radius of the replacement tool with M108", Page 1359

Description of function

Effect

M101 takes effect at the start of the block.

In order to reset **M101**, program **M102**.

Application example

Refer to your machine manual.

The function of **M101** can vary depending on the individual machine tool.

11 TOOL CALL 5 Z S3000	; Tool call
12 M101	; Activate automatic tool change

The control exchanges the tools and activates **M101** in the next NC block. The **TIME2** column of the tool management contains the maximum age for the tool life at the time the tool is called. If, during machining, the current tool age in the column **CUR_TIME** exceeds this value, the control inserts the replacement tool at a suitable point in the NC program. This exchange takes place after no more than one minute, unless the control has not concluded the active NC block yet. A useful application of this function is for automated programs on unattended machines.

Input

If you define **M101**, the control continues the dialog and prompts you for **BT**. With **BT** you define the number of NC blocks by which the automatic tool change may be delayed (up to 100 blocks). The content of the NC blocks, such as the feed rate or distance moved, influences the time by which the tool change is delayed.

If you do not define **BT**, the control uses the value 1 or, if applicable, a default value defined by the machine manufacturer.

The value for **BT**, the tool life verification, and the calculation of the automatic tool change have an influence on the machining time.

11 M101 BT10	; Activate automatic tool change after no more than 10 NC blocks

Notes

NOTICE

Danger of collision!

During an automatic tool change with **M101**, the control always retracts the tool in the tool axis first. There is danger of collision when retracting tools for machining undercuts, such as side milling cutters or T-slot milling cutters!

- Use **M101** only for machining operations without undercuts
- Deactivate the tool change with **M102**

- If you want to reset the current age of a tool (e.g. after changing the indexable inserts), enter the value 0 in the **CUR_TIME** column of the tool management.

  Further information: "Tool management ", Page 297
- For indexed tools, the control does not apply any data from the main tool. You must define a replacement tool (with index, if necessary) in each table row in the tool management. If an indexed tool is worn and therefore disabled, this does not apply to all indices. This means, for example, that the main tool can still be used.

  Further information: "Indexed tool", Page 276
- The higher the value of **BT**, the smaller will be the effect of an extended program duration through **M101**. Please note that this will delay the automatic tool change!
- The **M101** miscellaneous function is not available for turning tools and in turning mode (option 50).

Notes on tool change

- The control performs the automatic tool change at a suitable point in the NC program.
- The control cannot perform the automatic tool change at the following points in a program.
  - During a machining cycle
  - If radius compensation with **RR** or **RL** is active
  - Directly after an **APPR** approach function
  - Directly before a **DEP** departure function
  - Directly before and after a chamfer with **CHF** or a rounding with **RND**
  - During a macro
  - During a tool change
  - Directly after the NC functions **TOOL CALL** or **TOOL DEF**
- If the machine manufacturer does not define otherwise, the control moves the tool after the tool change as follows:
  - If the target position in the tool axis is below the current position, the tool axis is positioned last.
  - If the target position in the tool axis is above the current position, the tool axis is positioned first.

Notes on the input value BT

- To calculate a suitable initial value for **BT**, use the following formula:

  $BT = 10 \div t$

  t: average machining time of an NC block in seconds

  Round the result up to an integer value. If the calculated result is greater than 100, use the maximum input value of 100.
- In the optional machine parameter **M101BlockTolerance** (no. 202206) the machine manufacturer defines the standard value for the number of NC blocks by which the automatic tool change may be delayed. This standard value applies if you do not define **BT**.

Definition

Abbreviation	Definition
BT (block tolerance)	Number of NC blocks by which a tool change may be delayed.

23.5.2 Permitting positive tool oversizes with M107 (option 9)

Application

With **M107** (option 9) the control does not interrupt the machining process upon a positive delta value. The function is in effect with active 3D tool compensation and for **LN** straight lines.

Further information: "3D tool compensation (option 9)", Page 1126

With **M107** you can, for example, use the same tool in a CAM program for pre-finishing with oversize and then later for final finishing without oversize.

Further information: "Output formats of NC programs", Page 1302

Requirement

- Advanced Functions Set 2 (software option 9)

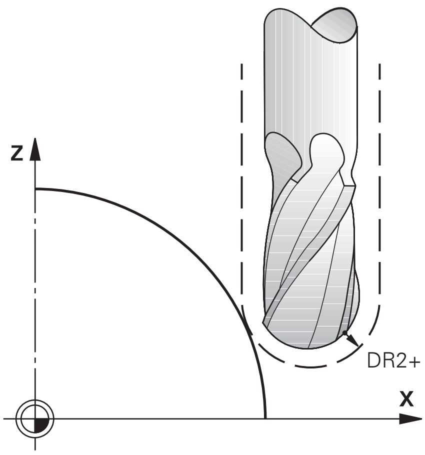
Description of function

Effect

M107 takes effect at the start of the block.

In order to reset **M107**, program **M108**.

Application example



11 TOOL CALL 1 Z S5000 DR2:+0.3	; Insert a tool with a positive delta value
12 M107	; Permit positive delta values

The control exchanges the tools and activates **M107** in the next NC block. That way the control permits positive delta values and does not issue an error message, such as during pre-finishing.

Without **M107** the control issues an error message upon positive delta values.

Notes

- Before actual machining, check in the NC program to make sure that the positive delta values of the tool will not result in contour damages or collisions.
- With peripheral milling the control issues an error message in the following case:

  $DR_{Tab} + DR_{Prog} > 0$

  Further information: "3D tool compensation during peripheral milling (option 9)", Page 1137
- With face milling the control issues an error message in the following cases:
  - $DR_{Tab} + DR_{Prog} > 0$
  - $R2 + DR2_{Tab} + DR2_{Prog} > R + DR_{Tab} + DR_{Prog}$
  - $R2 + DR2_{Tab} + DR2_{Prog} > 0$
  - $DR2_{Tab} + DR2_{Prog} > 0$

  Further information: "3D tool compensation during face milling (option 9)", Page 1130

Definition

Abbreviation	Definition
R	Tool radius
R2	Corner radius
DR	Delta value of the tool radius
DR2	Delta value of the corner radius
TAB	Value refers to the tool management
PROG	Value refers to the NC program, meaning from the tool call or from compensation tables

23.5.3 Checking the radius of the replacement tool with M108

Application

If you program **M108** before inserting a replacement tool, the control checks the replacement tool for any radius deviations.

Further information: "Automatically inserting a replacement tool with M101", Page 1355

Description of function

Effect

M108 takes effect at the end of the block.

Application example

11 TOOL CALL 1 Z S5000	; Insert the tool
12 M101 M108	; Activate automatic tool change and radius checking

The control exchanges the tool and activates the automatic tool change and radius checking in the next NC block.

If the maximum tool age of the tool expires during machining, the control inserts the replacement tool. The control checks the tool radius of the replacement tool based on the **M108** miscellaneous function defined previously. If the radius of the replacement tool is greater than the radius of the tool being replaced, the control issues an error message.

Without **M108** the control will not check the radius of the replacement tool.

Note

M108 is also used to reset **M107** (option 9).

Further information: "Permitting positive tool oversizes with M107 (option 9)", Page 1357

23.5.4 Suppressing touch probe monitoring with M141

Application

In conjunction with the touch probe cycles **3 MEASURING** or **4 MEASURING IN 3-D**, if the stylus is deflected, you can retract the touch probe in a positioning block with **M141**.

Description of function

Effect

M141 is in effect blockwise for straight lines and takes effect at the start of the block.

Application example

11 TCH PROBE 3.0 MEASURING	
12 TCH PROBE 3.1 Q1	
13 TCH PROBE 3.2 Y ANGLE: +0	
14 TCH PROBE 3.3 ABST +10 F100	
15 TCH PROBE 3.4 ERRORMODE1	
16 L IX-20 R0 F500 M141	; Retract with **M141**

In Cycle **3 MEASURING** the control probes the X axis of the workpiece. Since no retraction distance **MB** is defined in this cycle, the touch probe stands still after the deflection.

In NC block **16** the control retracts the touch probe against the probing direction by 20 mm. **M141** suppresses monitoring of the touch probe.

Without **M141** the control issues an error message as soon as you move the machine axes.

Further information: "Cycle 3 MEASURING ", Page 1832

Further information: "Cycle 4 MEASURING IN 3-D ", Page 1834

Note

NOTICE

Danger of collision!

The miscellaneous function **M141** suppresses the corresponding error message if the stylus is deflected. The control does not perform an automatic collision check with the stylus. Based on these two types of behavior, you must check whether the touch probe can retract safely. There is a risk of collision if you choose the wrong direction for retraction.

- Carefully test the NC program or program section in the **Program run, single block** operating mode

24

Variable Programming

24.1 Overview of variable programming

The control provides the following options for variable programming in the **FN** folder of the **Insert NC function** window:

Function group	Further information
Basic arithmetic operations	Page 1374
Trigonometric functions	Page 1376
Circle calculations	Page 1378
Jump commands	Page 1380
Special functions	Page 1381 Page 1394
SQL statements	Page 1417
String functions	Page 1400
Counters	Page 1409
Calculations using formulas	Page 1397
Function for the definition of complex contours	Page 410

24.2 Variables: Q, QL, QR and QS parameters

24.2.1 Basics

Application

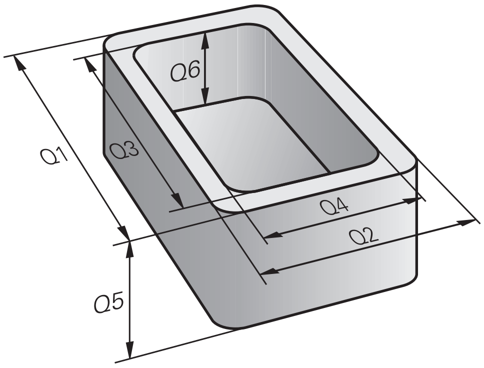
You can use the Q, QL, QR and QS parameters of the control, also referred to as variables, to take measurement results into account dynamically within calculations while machining.

For instance, you can program the following syntax elements variably:

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

This means that the same NC program can be used for different workpieces and values have to be changed in only one central place.

Description of function



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

For each variable type, you can define the variable range that the control displays on the **QPARA** tab of the **Status** workspace.

Further information: "Defining the contents of the QPARA tab", Page 191

Variable types

The control provides the following variables for numerical values:

- Q parameters
  Further information: "Q parameters", Page 1364
- QL parameters
  Further information: "QL parameters", Page 1364
- QR parameters
  Further information: "QR parameters", Page 1364

In addition, the control provides QS parameters for alpha-numeric values (e.g., texts).

Further information: "QS parameters", Page 1364

Q parameters

Q parameters affect all NC programs in the control's memory.

Q parameters have a local effect within macros and machine manufacturer cycles. This means that the control will not return changes to the NC program.

The control provides the following Q parameters:

Variable range	Meaning
0 to 99	User-defined Q parameters, if there are no overlaps with the HEIDENHAIN SL 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.

The control provides the following QL parameters:

Variable range	Meaning
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.

The control provides the following QR parameters:

Variable range	Meaning
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)

QS parameters

QS parameters affect all NC programs in the control's memory.

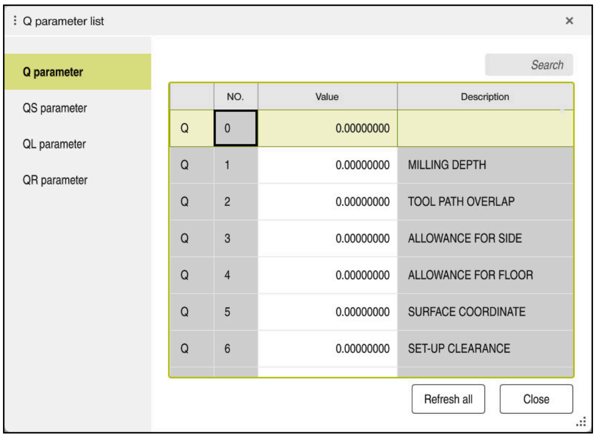
QS parameters have a local effect within macros and the machine manufacturer cycles. This means that the control will not return changes to the NC program.

The control provides the following QS parameters:

Variable range	Meaning
0 to 99	User-defined QS parameters, if there are no overlaps with the HEIDENHAIN SL 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

Q parameter list window

In the **Q parameter list** window, you can view and edit the values of all variables.



Q parameter list window, showing the Q parameter values

In the left-hand panel, you can select the variable type to be displayed.

The control displays the following information:

- Variable type (e.g., Q parameter)
- Number of the variable
- Value of variable
- Description in case of pre-assigned variables

If the cell in the **Value** column is displayed with a white background, you can edit its value.

> While the control is executing an NC program, you cannot edit the variables using the **Q parameter list** window. Changes are only possible while a program run has been interrupted or aborted.
>
> **Further information:** "Status overview on the TNC bar", Page 169
>
> This status is reached after an NC block has been executed, for example in the **Single Block** mode
>
> 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.
>
> **Further information:** "Variable types", Page 1364

The following search options are available in the **Q parameter list** window:

- Search the entire table for any strings
- Search the **NR** column for a unique variable number

Further information: "Searching the Q parameter list window", Page 1367

You can open the **Q parameter list** window in the following operating modes:

- **Editor**
- **Manual**
- **Program Run**

In the **Manual** and **Program Run** operating modes, the window can be opened with the **Q** key.

Searching the Q parameter list window

To search the **Q parameter list** window:

- Select any cell with a gray background
- Enter the desired string
- > The control opens an input field and searches the column of the selected cell for this string.
- > The control marks the first result that starts with the search string.
- Select the next result, if necessary

The control displays an input field above the table. Alternatively, you can use this input field to navigate to a unique variable number. To select the input field, press the **GOTO** key.

Notes

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: "Preassigned Q parameters", Page 1368

- You can enter fixed and variable values mixed in the NC program.
- You can assign a maximum of 255 characters to QS parameters.
- You can use the **Q** key to create an NC block to assign a value to a variable. If you press the key again, the control changes the variable type in the order **Q**, **QL**, **QR**.

  On the virtual keyboard, this procedure only works with the **Q** key in the NC functions area.

  Further information: "Virtual keyboard of the control bar", Page 1508
- 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 reset variables to the **Undefined** status. For example, if you program a position using an undefined Q parameter, the control ignores this movement.

  Further information: "Assigning the Undefined status to a variable", Page 1376
- 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.

Notes on QR parameters and backup

The control saves QR parameters within a backup.

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.

Further information: "Backup and restore", Page 2148

24.2.2 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.

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

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

Further information: "Tool data", Page 275

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

Further information: "Designation of the axes on milling machines", Page 208

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

Further information: "Miscellaneous Functions", Page 1317

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**.

Further information: "Cycles for milling", Page 507

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 with **PGM CALL**, the control uses 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.

Further information: "Tool data", Page 275

Calculated coordinates of the rotary axes: Q120 to Q122

The control assigns the calculated coordinates of the rotary axes to the Q parameters **Q120** to **Q122**:

Q parameters	Rotary axis coordinates
Q120	**AXIS ANGLE IN THE A AXIS**
Q121	**AXIS ANGLE IN THE B AXIS**
Q122	**AXIS ANGLE IN THE C AXIS**

Measurement results from touch-probe cycles

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

> The help graphics of the touch-probe cycles show whether the control saves a measurement result in a variable or not.
>
> **Further information:** "Help workspace", Page 1506

Further information: "Programmable Touch Probe Cycles", Page 1589

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.

Q parameters Q115 to Q119

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

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

Q parameters Q150 to Q160

The control assigns the measured actual values to the Q parameters **Q150** to **Q160**:

Q parameters	Measured actual values
Q150	**MEASURED ANGLE**
Q151	**ACTL. VALUE, REF AXIS**
Q152	**ACTL.VALUE, MINOR AXIS**
Q153	**ACTUAL VALUE, DIAMETER**
Q154	**ACT.VAL. PCKT REF AX.**
Q155	**ACT.VAL. PKT MINOR AX.**
Q156	**ACTUAL VALUE OF LENGTH**
Q157	**ACTL.VAL., CENTERLINE**
Q158	**Projectd. angle A axis**
Q159	**Projectd. angle B axis**
Q160	**COORD., MEASURING AXIS** Coordinate of the axis selected in the cycle

Q parameters Q161 to Q167

The control assigns the calculated deviation values to the Q parameters **Q161** to **Q167**:

Q parameters	Calculated deviation
Q161	ERROR, CENTR, REF AX. Deviation of center in main axis
Q162	ERROR, CENTR, MINOR AX Deviation of center in the secondary axis
Q163	ERROR OF DIAMETER
Q164	ERROR, PCKT., REF AX. Deviation of pocket length in the main axis
Q165	ERROR, CENTR, MINOR AX Deviation of pocket width in the secondary axis
Q166	ERROR OF LENGTH Deviation of the measured length
Q167	ERROR OF CENTERLINE Deviation of the centerline position

Q parameters Q170 to Q172

The control assigns the determined spatial angle values to the Q parameters **Q170** to **Q172**:

Q parameters	Determined spatial angles
Q170	SPATIAL ANGLE A
Q171	SPATIAL ANGLE B
Q172	SPATIAL ANGLE C

Q parameters Q180 to Q182

The control assigns the determined workpiece status to the Q parameters **Q180** to **Q182**:

Q parameters	Workpiece status
Q180	WORKPIECE IS GOOD
Q181	WORKPIECE NEEDS REWORK
Q182	WORKPIECE IS SCRAP

Q parameters Q190 to Q192

The control reserves the Q parameters **Q190** to **Q192** for the results of tool measurements with a laser measuring system.

Q parameters Q195 to Q198

The control reserves the Q parameters **Q195** to **Q198** for internal use:

Q parameters	Reserved for internal use
Q195	**MARKER FOR CYCLES**
Q196	**MARKER FOR CYCLES**
Q197	**MARKER FOR CYCLES** Cycles with position pattern
Q198	**NO., LAST TCH-PRB CYC** Number of the last active touch-probe cycle

Q parameter Q199

The value of the Q parameter **Q199** depends on the status of tool measurement with a tool touch probe:

Q parameters	Status of tool measurement with a tool touch probe
Q199 = 0.0	Tool is within tolerance
Q199 = 1.0	Tool is worn (**LTOL**/**RTOL** is exceeded)
Q199 = 2.0	Tool is broken (**LBREAK**/**RBREAK** is exceeded)

Q parameters Q950 to Q967

The control assigns the measured actual values resulting from the **14xx** touch-probe cycles to the Q parameters **Q950** to **Q967**:

Q parameters	Measured actual values
Q950	**P1 measured main axis**
Q951	**P1 measured minor axis**
Q952	**P1 measured tool axis**
Q953	**P2 measured main axis**
Q954	**P2 measured minor axis**
Q955	**P2 measured tool axis**
Q956	**P3 measured main axis**
Q957	**P3 measured minor axis**
Q958	**P3 measured tool axis**
Q961	**Measured SPA** Spatial angle **SPA** in the working plane coordinate system **WPL-CS**
Q962	**Measured SPB** Spatial angle **SPB** in the **WPL-CS**
Q963	**Measured SPC** Spatial angle **SPC** in the **WPL-CS**
Q964	**Meas. basic rotation** Rotational angle in the input coordinate system **I-CS**
Q965	**Meas. table rotation**
Q966	**Measured diameter 1**
Q967	**Measured diameter 2**

Q parameters Q980 to Q997

The control assigns the calculated deviations resulting from the **14xx** touch-probe cycles to the Q parameters **Q980** to **Q997**:

Q parameters	Measured deviations
Q980	**P1 error main axis**
Q981	**P1 error minor axis**
Q982	**P1 error tool axis**
Q983	**P2 error main axis**
Q984	**P2 error minor axis**
Q985	**P2 error tool axis**
Q986	**P3 error main axis**
Q987	**P3 error minor axis**
Q988	**P3 error tool axis**
Q994	**Error: basic rotation** Angle in the input coordinate system **I-CS**
Q995	**Meas. table rotation**
Q996	**Error: diameter 1**
Q997	**Error: diameter 2**

Q parameter Q183

The value of the Q parameter **Q183** depends on the workpiece status as measured by the 14xx touch-probe cycles:

Q parameters	Workpiece status
Q183 = –1	Not defined
Q183 = 0	Pass
Q183 = 1	Rework
Q183 = 2	Scrap

24.2.3 Basic arithmetic folder

Application

In the **Basic arithmetic** folder of the **Insert NC function** window, the control offers the functions **FN 0** to **FN 5**.

You can assign numerical values to variables using the **FN 0** function. You then use a variable instead of the fixed number in the NC program. You can also use preassigned variables (e.g., the active tool radius **Q108**). Using the functions **FN 1** to **FN 5**, you can make calculations with the variable values in your NC program.

Related topics

- Preassigned variables

  Further information: "Preassigned Q parameters", Page 1368
- Programmable touch probe cycles

  Further information: "Programmable Touch Probe Cycles", Page 1589
- Calculations using formulas

  Further information: "Formulas in the NC program", Page 1397

Description of function

The **Basic arithmetic** folder contains the following functions:

Icon	Function
=	**FN 0**: Assignment Example: **FN 0: Q5 = +60** Q5 = 60 Assign a value or the **Undefined** status
+	**FN 1**: Addition Example: **FN 1: Q1 = -Q2 + -5** Q1 = –Q2+(–5) Calculate and assign the sum of two values
–	**FN 2**: Subtraction Example: **FN 2: Q1 = +10 - +5** Q1 = +10–(+5) Calculate and assign the difference of two values.
×	**FN 3**: Multiplication Example: **FN 3: Q2 = +3 * +3** Q2 = 3*3 Calculate and assign the product of two values.
/	**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
√	**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

To the left of the equal sign, define the variable to which the result should be assigned.

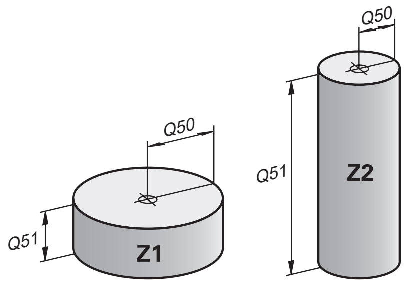
To the right of the equal sign, you can use fixed or variable values. The variables and numerical values in the equations can be entered with an algebraic sign.

Part families

For part families, for example, you can program the characteristic workpiece dimensions as variables. When machining the individual workpieces, assign a numerical value to each variable.

11 LBL "Z1"	
12 FN 0: Q50 = +30	; Assign the value **30** to the cylinder radius **Q50**
13 FN 0: Q51 = +10	; Assign the value **10** to the cylinder height **Q51**
*** - ...**	
21 L X +Q50	; Result corresponds to **L X +30**

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

Assigning the Undefined status to a variable

To assign the **Undefined** status to a variable:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **FN 0**
- Enter the number of the variable (e.g., **Q5**)
- Select **SET UNDEFINED**
- Confirm your input
- > The control assigns the **Undefined** status to the variable.

Notes

- The control distinguishes between undefined variables and variables with the value 0.
- You cannot divide by 0 (**FN 4**).
- You cannot extract a square root from a negative value (**FN 5**).

24.2.4 Trigonometric functions folder

Application

In the **Trigonometric functions** folder of the **Insert NC function** window, the control provides the functions **FN 6** to **FN 8** and **FN 13**.

You can use these functions to calculate trigonometric functions for purposes such as programming variable triangular contours.

Description of function

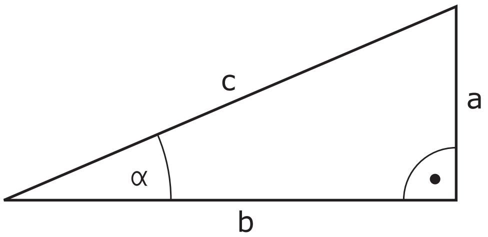
The **Trigonometric functions** folder contains the following functions:

Icon	Function
SIN	**FN 6**: Sine Example: **FN 6: Q20 = SIN -Q5** Q20 = sin(–Q5) Calculate and assign the sine of an angle in degrees
COS	**FN 7**: Cosine Example: **FN 7: Q21 = COS -Q5** Q21 = cos(–Q5) Calculate and assign the cosine of an angle in degrees
LEN	**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).
ANG	**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°)

To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal sign, you can use fixed or variable values. The variables and numerical values in the equations can be entered with an algebraic sign.

Definition



Side or trigonometric function	Meaning
a	Opposite side The side opposite to angle **α**
b	Adjacent side The side adjacent to angle **α**
c	Hypotenuse The longest side of the triangle, opposite to the right angle
Sine	sin **α** = opposite side/hypotenuse sin **α** = a/c
Cosine	cos **α** = adjacent side/hypotenuse cos **α** = b/c
Tangent	tan **α** = opposite side/adjacent side tan **α** = a/b or tan **α** = sin **α**/cos **α**
Arc tangent	**α** = arctan(a/b) or **α** = arctan(sin **α**/cos **α**)

Example

a = 25 mm

b = 50 mm

α = arctan(a/b) = arctan 0.5 = 26.57°

Furthermore:

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

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

11 Q50 = ATAN (+25 / +50)	Calculate angle α
12 FN 8: Q51 = +25 LEN +50	Calculate side length c

24.2.5 Circle calculation folder

Application

In the **Circle calculation** folder of the **Insert NC function** window, the control provides the functions **FN 23** and **FN 24**.

These functions allow you to calculate the center of a circle and the radius of the circle based on the coordinates of three or four points on the circle (e.g., the position and size of a circle segment).

Description of function

The **Circle calculation** folder contains the following functions:

Icon	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**.
	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**.

To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal sign, define the variable starting from which the control is to determine the circle data from the next variables.

The coordinates of the circle data are stored in successive variables. These coordinates must be in the working plane. You must save the coordinates of the main axis before the coordinates of the secondary axis (e.g., **X** before **Y** for tool axis **Z**).

Further information: "Designation of the axes on milling machines", Page 208

Application example

11 FN 23: Q20 = CDATA Q30	; Circle calculation with three points on the circle

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**

> NC function **FN 24** uses four pairs of coordinate values and thus eight successive Q parameters.

Note

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.

24.2.6 Jump commands folder

Application

In the **Jump commands** folder of the **Insert NC function** window, the control provides the functions **FN 9** to **FN 12** for jumps with if-then decisions.

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.

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

Related topics

- Jumps without condition with **CALL LBL** label call

  Further information: "Subprograms and program section repeats with the label LBL", Page 384

Description of function

The **Jump commands** folder contains the following functions for if-then decisions:

Icon	Function
=	**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.
	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.
	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.
≠	**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.
>	**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.
<	**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.

You can enter fixed or variable values for if-then decisions.

Unconditional jump

Unconditional jumps are jumps whose condition is always fulfilled.

11 FN 9: IF+0 EQU+0 GOTO LBL1	; Unconditional jump with **FN 9** whose condition is always fulfilled

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.

Further information: "Subprograms", Page 386

Definitions

Abbreviation	Definition
IF	If
EQU (equal)	Equal to
NE (not equal)	Not equal to
GT (greater than)	Greater than
LT (less than)	Less than
GOTO (go to)	Go to
UNDEFINED	Undefined
DEFINED	Defined

24.2.7 Special functions for programming with variables

Output error messages with FN 14: ERROR

Application

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

Related topics

- Error numbers pre-assigned by HEIDENHAIN
  Further information: "Preassigned error numbers for FN 14: ERROR", Page 2265
- Error messages in the notification menu
  Further information: "Message menu on the information bar", Page 1532

Description of function

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.

You define the error number for the desired error message.

The error numbers are grouped as follows:

Error number range	Error message
0 ... 999	Machine-dependent dialog
1000 ... 1199	Control-dependent dialog

Further information: "Preassigned error numbers for FN 14: ERROR", Page 2265

Input

11 FN 14: ERROR=1000	; Output error message with **FN 14**

Insert NC function ▶ **All functions** ▶ **FN** ▶ **Special functions** ▶ **FN 14 ERROR**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 14: ERROR	Start of syntax for error message output
1000	Number of the error message Fixed or variable number

Note

Please be aware that not all error messages might be available, depending on the control and the software version.

Outputting text formatted with FN 16: F-PRINT

Application

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

Related topics

- Automatically generated measurement log for touch probe cycles
  Further information: "Recording the results of measurement", Page 1774
- Print to a connected printer
  Further information: "Printers", Page 2130

Description of function

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.

Further information: "Source file for content and formatting", Page 1382

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 source file
  Further information: "Keywords", Page 1384

Source file for content and formatting

Define the formatting and the content of the output file in a source file with the extension ***.a**.

Formatting

The formatting of the source file can be defined with the following formatting characters:

> Please note that the input is case-sensitive.

Formatting characters	Function
“...“	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)
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 source file line (e.g., data type and variable)
;	End of the source file line
*****	Initiate a comment line within the source 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

Keywords

You can define the contents of the output file with the following keywords:

Keyword	Function
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
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

Keyword	Function
L_SLOVAK	Outputs text only for Slovakian conversational language
L_TURKISH	Outputs text only for Turkish conversational language
L_ALL	Display text independently of the conversational language
HOUR	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

Input

11 FN 16: F-PRINT TNC:\mask.a / TNC:\Prot1.txt	; Output file **Prot1.txt** with the source from **Mask.a**

To navigate to this function:

Insert NC function ▶ **FN** ▶ **Special functions** ▶ **FN 16 F-PRINT**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 16: F-PRINT	Start of syntax for formatted output of contents
***.a**	Path of the source file for the output format
/	Separator between the two paths
TNC:\Prot1.txt	Path under which the control saves the output file Fixed or variable name The file name extension of the log file determines the file type of the output (e.g., TXT, A, XLS, HTML).

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

Output options

Screen output

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 you want to replace the content of the window for multiple screen outputs in the NC program, define the **M_CLOSE** or **M_TRUNCATE** keyword.

The control opens the **FN16-PRINT** window for screen output. The window remains open until you close it. While the window is open, you can operate the control in the background and change to another operating mode.

You can close the window in the following ways:

- **OK** button
- Defining the **SCLR:** output path (Screen Clear)

Saving the output file

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 the output file

You can use the **FN 16** function to print output files to a connected printer.

Further information: "Printers", Page 2130

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**

Notes

- Use the optional machine parameters **fn16DefaultPath** (no. 102202) and **fn16DefaultPathSim** (no. 102203) to define a path under which the control saves the output files.

  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 only define the file name as the target path of the output file in the FN function, the control saves the output file in the folder of the NC program.
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- If you specify the **%RS** function in the source file, the control takes over the defined content without formatting. This allows you to output a path specification with QS parameters, for example.
- In the settings of the **Program** workspace, you can specify whether the control displays a screen output in a window.

  If you deactivate the screen output, the control will not display a window. The control will display the contents anyway on the **FN 16** tab of the **Status** workspace.

  Further information: "Settings in the Program workspace", Page 220

  Further information: "FN16 tab", Page 173

Example

Example of a source file that generates an output file with variable content:

“TOUCHPROBE“;

“%S“,QS1;

M_EMPTY_HIDE;

“%S“,QS2;

“%S“,QS3;

M_EMPTY_SHOW;

“%S“,QS4;

"DATE: %02d.%02d.%04d",DAY,MONTH,YEAR4;

"TIME: %02d:%02d",HOUR,MIN;

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**:

FN16-PRINT window

Read system data with FN 18: SYSREAD

Application

The **FN 18: SYSREAD** function can be used to read system data and store this data in variables.

Related topics

- List of the system data of the control

  Further information: "List of FN functions", Page 2271
- Read system data using QS parameters

  Further information: "Read system data with SYSSTR", Page 1402

Description of function

The control always outputs system data in the metric system with **FN 18: SYSREAD**, regardless of the unit of the NC program.

Input

11 FN 18: SYSREAD Q25 = ID210 NR4 IDX3	; Save the active dimension factor of the Z axis in **Q25**

To navigate to this function:

Insert NC function ▶ **FN** ▶ **Special functions** ▶ **FN 18 SYSREAD**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN18: SYSREAD	Read the syntax initiator for system data
Q/QL/QR or **QS**	Variable in which the control stores the information Fixed or variable number or name
ID	Group number of the system datum Fixed or variable number or name
NR	System data number Fixed or variable number or name Optional syntax element
IDX	Index Fixed or variable number or name Optional syntax element
.	Sub-index for system data for tools Fixed or variable number or name Optional syntax element

Note

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: "Reading table values with TABDATA READ", Page 1992

Transfer values to PLC with FN 19: PLC

Application

The **FN 19: PLC** function transfers up to two fixed or variable values to the PLC.

Description of function

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

Synchronizing NC and PLC with FN 20: WAIT FOR

Application

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.

Description of function

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 **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**.

Application example

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**

In this example, you stop the internal pre-calculation of the control in order to determine the current position of the X axis.

Transferring values to PLC with FN 29: PLC

Application

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

Description of function

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

Creating your own cycles with FN 37: EXPORT

Application

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

Description of function

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

Sending information from the NC program with FN 38: SEND

Application

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).

Description of function

Data is transferred via a TCP/IP connection.

For more detailed information, consult the RemoTools SDK manual.

Input

11 FN 38: SEND /"Q-Parameter Q1: %F Q23: %F" / +Q1 / +Q23	; Write values from **Q1** and **Q23** to the logbook

To navigate to this function:

Insert NC function ▶ **FN** ▶ **Special functions** ▶ **FN 38 SEND**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 38: SEND	Send syntax initiator for information
"...", **QS**	Format of the text to be transmitted Fixed or variable name Output text with up to seven placeholders for the values of the variables (e.g., **%F**) **Further information:** "Source file for content and formatting", Page 1382
/	Contents of the up to seven placeholders in the output text Fixed or variable number Optional syntax element

Notes

- Both fixed and variable numbers and texts are case-sensitive, so enter them correctly.
- 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

11 FN 38: SEND /"JOB:1234_STEP:1_CREATE"	; Create job
12 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
13 FN 38: SEND /"JOB:1234_STEP:1_START"	; Start job
14 FN 38: SEND /"JOB:1234_STEP:1_PREPARATION"	; Start preparation
15 FN 38: SEND /"JOB:1234_STEP:1_PRODUCTION"	; Production
16 FN 38: SEND /"JOB:1234_STEP:1_STOP"	; Stop job
17 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.

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

24.2.8 NC functions for freely definable tables

Opening a freely definable table with FN 26: TABOPEN

Application

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**.

Related topics

- Content and creation of freely definable tables
  Further information: "Freely definable tables", Page 2034
- Access to table values in case of low computing power
  Further information: "Table access with SQL statements", Page 1417

Description of function

Select the freely definable table to be opened by entering its path. Enter the file name with the ***.tab** extension.

Input

```
11 FN 26: TABOPEN TNC:\table\AFC.TAB    ; Open table with FN 26
```

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ FN 26 TABOPEN

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 26: TABOPEN	Start of syntax for opening a table
TNC:\table \AFC.TAB	Path of the table to be opened Fixed or variable name

Note

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.

Writing to a freely definable table with FN 27: TABWRITE

Application

With the **FN 27: TABWRITE** NC function, you write to the table that you previously opened with **FN 26: TABOPEN**.

Related topics

- Contents and creation of freely definable tables
  Further information: "Freely definable tables", Page 2034
- Opening a freely definable table
  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 1394

Description of function

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. The content to be written to the columns must have been defined previously, using variables.

Input

11 FN 27: TABWRITE 2/“Length,Radius“ = Q2	; Write to table with **FN 27**

Insert NC function ▶ **All functions** ▶ **FN** ▶ **Special functions** ▶ **FN 27 TABWRITE**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 27: TABWRITE	Start of syntax for writing to a table
2	Row number of the table to be written to Fixed or variable number
“Length,Radius“	Column names in the table to be written to Fixed or variable name Use commas to separate multiple column names.
Q2	Variable for the contents to be written

Notes

- 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.

Example

11 Q5 = 3.75	; Define the value for the **Radius** column
12 Q6 = -5	; Define the value for the **Depth** column
13 Q7 = 7.5	; Define the value for the **D** column
14 FN 27: TABWRITE 5/“Radius,Depth,D“ = Q5	; Write defined values to the table

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.

Reading a freely definable table with FN 28: TABREAD

Application

With the **FN 28: TABREAD** NC function, you can read data from the table previously opened with **FN 26: TABOPEN**.

Related topics

- Content and creation of freely definable tables

  Further information: "Freely definable tables", Page 2034
- Opening a freely definable table

  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 1394
- Writing a freely definable table

  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 1394

Description of function

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.

Input

11 FN 28: TABREAD Q1 = 2 / "Length"	; Read table with **FN 28**

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ FN 28 TABREAD

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 28: TABREAD	Start of syntax for reading from a table
Q1	Variable for the source text The control uses this variable to save the contents from the table cells to be read.
2	Row number in the table to be read Fixed or variable number
"Length"	Column name in the table to be read Fixed or variable name Use commas to separate multiple column names.

Note

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**).

Example

11 FN 28: TABREAD Q10 = 6/“X,Y,D“	; Read numeric values from columns **X**, **Y** and **D**
12 FN 28: TABREAD QS1 = 6/“DOC“	; Read the alphanumeric value from the **DOC** column

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.

24.2.9 Formulas in the NC program

Application

With the **Formula Q/QL/QR** NC function, you can define multiple arithmetic operations in a single NC block using fixed or variable values. You can also assign a single value to a variable.

Related topics

- String formula for strings
  Further information: "String functions", Page 1400
- Define a single calculation in an NC block
  Further information: "Basic arithmetic folder", Page 1374

Description of function

As the first entry, you define the variable to which you assign the result.

To the right of the equal sign, define the arithmetic operations or a value that the control assigns to the variable.

When defining the **Formula Q/QL/QR** NC function, you can open a virtual keyboard for formula input containing all available arithmetic operators in the action bar or form. The virtual keyboard also has a formula input mode.

Further information: "Virtual keyboard of the control bar", Page 1508

Rules for formulas

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).

Further information: "Example", Page 1400

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	**+**, **-**

Further information: "Arithmetic operations", Page 1398

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

Arithmetic operations

The virtual keyboard for formula input allows you to perform the following arithmetic operations:

Button	Arithmetic operation	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
( )	**Parenthesize** Example: **Q12 = Q1 * (Q2 + Q3)**	Expression in 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

Button	Arithmetic operation	Operator
^ **^**	**Exponentiation** Example: **Q15 = 3 ^ 3**	Power
PI **PI**	**Use the "pi" constant** π = 3.14159 Example: **Q15 = PI**	
LN **LN**	**Calculate the natural logarithm (LN)** Base = e = 2.7183 Example: **Q15 = LN Q11**	Function
LOG **LOG**	**Calculate the logarithm** Base = 10 Example: **Q33 = LOG Q22**	Function
EXP **EXP**	**Use the exponential function (e ^ n)** Base = e = 2.7183 Example: **Q1 = EXP Q12**	Function
NEG **NEG**	**Negate** Multiply by –1 Example: **Q2 = NEG Q1**	Function
INT **INT**	**Calculate an integer** Truncate decimal places Example: **Q3 = INT Q42** The **INT** function does not round off—it simply truncates the decimal places. Input: **0...999999999**	Function
ABS **ABS**	**Calculate the absolute value** Example: **Q4 = ABS Q22**	Function
FRAC **FRAC**	**Calculate a fraction** Truncate the digits before the decimal point Example: **Q5 = FRAC Q23**	Function
SGN **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

Further information: "Basic arithmetic folder", Page 1374
Further information: "Trigonometric functions folder", Page 1376

You can also define arithmetic operations for strings.

Further information: "String functions", Page 1400

Example

Multiplication and division before addition and subtraction

11 Q1 = 5 * 3 + 2 * 10	; Result = 35

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

Power before addition and subtraction

11 Q2 = SQ 10 - 3^3	; Result = 73

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

Function before power

11 Q4 = SIN 30 ^ 2	; Result = 0.25

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

Brackets before function

11 Q5 = SIN (50 - 20)	; Result = 0.5

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

24.3 String functions

Application

The string functions allow you to define and process strings using QS parameters (e.g., in order to create variable logs with **FN 16: F-PRINT**). In computing, a string designates an alphanumerical sequence of characters.

Related topics

- Ranges of variables

  Further information: "Variable types", Page 1364

Description of function

You can assign up to 255 characters to a QS parameter.

The following characters are permitted within QS parameters:

- Characters
- Numbers
- Special characters, for example **?**
- Control characters, for example **** for paths
- Spaces

The individual string functions are programmed using the free syntax input.

Further information: "Editing NC functions", Page 230

The values of QS parameters can be processed or checked with the **Formula Q/QL/QR** and **String formula QS** NC functions.

Syntax	NC function	Higher-level NC function
DECLARE STRING	Assign an alphanumeric value to a QS parameter **Further information:** "Assigning an alphanumeric value to a QS parameter", Page 1404	
STRING FORMULA	Concatenate contents of QS parameters and assign them to a QS parameter **Further information:** "Concatenation of alphanumeric values", Page 1405	**String formula QS**
TONUMB	Convert the alphanumeric value of a QS parameter to a numerical value and assign it to a Q, QL, or QR parameter **Further information:** "Converting alphanumeric values to numerical values ", Page 1405	**Formula Q/QL/QR**
TOCHAR	Convert a numerical value to an alphanumeric value and assign it to a QS parameter **Further information:** "Converting numerical values to alphanumeric values", Page 1405	**String formula QS**
SUBSTR	Copy a substring from a QS parameter and assign it to a QS parameter **Further information:** "Copying a substring from a QS parameter", Page 1406	**String formula QS**
SYSSTR	Read system data and assign the contents to a QS parameter **Further information:** "Read system data with SYSSTR", Page 1402	**String formula QS**
INSTR	Search for a substring in a QS parameter and assign the retrieved characters to a Q, QL, or QS parameter **Further information:** "Searching for a substring within QS parameter contents", Page 1406	**Formula Q/QL/QR**
STRLEN	Determine the string length of a QS parameter and assign it to a Q, QL, or QR parameter **Further information:** "Determining the number of characters in QS parameter contents", Page 1406	**Formula Q/QL/QR**
STRCOMP	Compare QS parameters in ascending lexical order and assign the result to a Q, QL, or QR parameter **Further information:** "Comparing the lexical order of two alphanumerical strings", Page 1407	**Formula Q/QL/QR**
CFGREAD	Read the content of a machine parameter and assign it to a QS parameter **Further information:** "Accepting the contents of a machine parameter", Page 1408	■ **String formula QS** ■ **Formula Q/QL/QR**

Read system data with SYSSTR

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**.

You can read the following system data:

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.
Kinematics, 10290	10	Kinematics programmed in the last **FUNCTION MODE** NC function

Group name, ID no.	Number	Meaning
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
Data for pallet machining, 10510	1	Name of the pallet being machined
	2	Path of the currently selected pallet table
NC software version, 10630	10	Number of the NC software version
Information for unbalance cycle, 10855	1	Path of the unbalance calibration table The unbalance calibration table is part of the active kinematics.
Tool data, 10950	1	Current tool name
	2	Content of the **DOC** column of the current tool
	3	AFC control settings of the current tool
	4	Tool-carrier kinematics of the current tool

Reading machine parameters with CFGREAD

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:

Symbol	Type	Meaning
	Key	Group name of the machine parameter The group name can be specified optionally
	Entity	Parameter object The name always begins with **Cfg**
	Attribute	Name of the machine parameter
	Index	List index of the machine parameter The list index can be specified optionally

> 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.

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.

Further information: "Accepting the contents of a machine parameter", Page 1408

24.3.1 Assigning an alphanumeric value to a QS parameter

Before you can use and process alphanumeric values, you need to assign characters to the QS parameters. Use the **DECLARE STRING** command to do so.

To assign an alphanumeric value to a QS parameter:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **DECLARE STRING**
- Define a QS parameter for the result
- Select **Name**
- Enter the desired value
- End the NC block
- Execute the NC block
- > The control saves the entered value in the target parameter.

In this example, the control assigns an alphanumeric value to the QS parameter **QS10**.

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

24.3.2 Concatenation of alphanumeric values

With the **||** concatenation operator, you can concatenate the contents of multiple QS parameters. This allows you to combine fixed and variable alphanumeric values.

To concatenate the contents of multiple QS parameters:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **String formula QS**
- Define a QS parameter for the result
- Open keyboard for formula input
- Select concatenation operator **||**
- To the left of the concatenation operator, specify the number of the QS parameter that contains the first substring
- To the right of the concatenation operator, define the number of the QS parameter that contains the second substring
- End NC block
- Confirm your input
- > The control saves the substrings after execution consecutively as an alphanumeric value in the target parameter.

In this example, the control concatenates the contents of the QS parameters **QS12** and **QS13**. The alphanumeric value is assigned to the QS parameter **QS10**.

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**

24.3.3 Converting alphanumeric values to numerical values

With the **TONUMB** NC function, you save exclusively numeric characters from a QS parameter to a different variable type. Then, you can use these values in calculations.

In this example, the control converts the alphanumeric value of the QS parameter **QS11** to a numerical value. This value is assigned to the Q parameter **Q82**.

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

24.3.4 Converting numerical values to alphanumeric values

With the **TOCHAR** NC function, you can save the content of a variable in a QS parameter. The saved content can, for example, be concatenated with other QS parameters.

In this example, the control converts the numerical value of the Q parameter **Q50** to an alphanumeric value. The control assigns this value to the QS parameter **QS11**.

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

24.3.5 Copying a substring from a QS parameter

With the **SUBSTR** NC function, you can save a defined substring from a QS parameter to another QS parameter. For example, you can use this NC function to extract the file name from an absolute file path.

In this example, the control saves a substring of the QS parameter **QS10** to the QS parameter **QS13**. Using the **BEG2** syntax element, you define that the control ignores the first two characters and starts copying from the third character. With the **LEN4** syntax element, you define that the control copies the next four characters.

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

24.3.6 Searching for a substring within QS parameter contents

With the **INSTR** NC function , you can check whether a particular substring is contained within a QS parameter. This allows you to determine, for example, whether the concatenation of multiple QS parameters was successful. For the check, you must indicate two QS parameters. The control searches the first QS parameter for the content of the second QS parameter.

If the substring is found, the control saves the number of characters until it reaches the reference of the substring to the result parameter. If multiple occurrences are found, the result is identical because the control saves the first one.

If the substring searched for is not found, the control saves the total number of characters in the result parameter.

In this example, the control searches the QS parameter **QS10** for the string saved in **QS13**. The search starts from the third character. When counting the characters, the control starts from zero. The control assigns the occurrence to the Q parameter **Q50** as a number of characters.

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

24.3.7 Determining the number of characters in QS parameter contents

The **STRLEN** NC function determines the number of characters in QS parameter contents. With this NC function, you can, for example, determine the length of a file path.

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

In this example, the control determines the number of characters in the QS parameter **QS15**. The numerical value of the number of characters is assigned to the Q parameter **Q52**.

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

24.3.8 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.

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).

In this example, the control compares the lexical order of **QS12** and **QS14**. The result is assigned to the Q parameter **Q52** as a numerical value.

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

24.3.9 Accepting the contents of a machine parameter

Depending on the content of the machine parameter, you can use the **CFGREAD** NC function to take over alphanumeric values to QS parameters or numerical values to Q, QL or QR parameters.

In this example, the control saves the overlap factor from the **pocketOverlap** machine parameter as a numerical value in a Q parameter.

Specified settings in the machine parameters:

- **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

The **CFGREAD** NC function contains the following syntax elements:

- **KEY_QS**: Group name (key) of the machine parameter

> It no group name is available, define a blank value for the corresponding QS parameter.

- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

Further information: "Reading machine parameters with CFGREAD", Page 1404

Note

If you use the **String Formula QS** NC function, the result is always an alphanumeric value. If you use the **Formula Q/QL/QR** NC function, the result is always a numerical value.

24.4 Defining counters with FUNCTION COUNT

Application

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.

Description of function

The counter reading remains the same after a restart of the control.

The control only takes the **FUNCTION COUNT** function into account in the **Program Run** operating mode.

The control shows the current counter value and the defined target number on the **PGM** tab of the **Status** workspace.

Further information: "PGM tab", Page 178

Input

11 FUNCTION COUNT TARGET5	; Set the target count of the counter to **5**

Insert NC function ▶ **All functions** ▶ **FN** ▶ **FUNCTION COUNT**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION COUNT	Syntax initiator for the counter
INC, **RESET**, **ADD**, **SET**, **TARGET** or **REPEAT**	Define counting function **Further information:** "Counting functions", Page 1409

Counting functions

The **FUNCTION COUNT** NC function provides the following counter functions:

Syntax	Function
INC	Increase the counter by 1
RESET	Reset the counter
ADD	Increase the counter by a defined value Fixed or variable number or name Input: **0...9999**
SET	Assign a defined value to the counter Fixed or variable number or name Input: **0...9999**
TARGET	Define the target count to be reached Fixed or variable number or name Input: **0...9999**
REPEAT	Repeat the NC program from the label if the defined target count has not been reached yet Fixed or variable number or name

Notes

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.

- The machine manufacturer uses the optional machine parameter **CfgNcCounter** (no. 129100) to define whether you can edit the counter.
- You can engrave the current counter reading with Cycle **225 ENGRAVING**.
  Further information: "Cycle 225 ENGRAVING ", Page 704

24.4.1 Example

11 FUNCTION COUNT RESET	; Reset counter value
12 FUNCTION COUNT TARGET10	; Define the target count of machining operations
13 LBL 11	; Set a jump label
*** - ...**	; Execute the machining operation
21 FUNCTION COUNT INC	; Increase the counter reading by 1
22 FUNCTION COUNT REPEAT LBL 11	; Repeat the machining operation until the target count has been reached

24.5 Program defaults for cycles

24.5.1 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 use **PREDEF** to simply reference the value defined at the beginning of the program.

The following **GLOBAL DEF** functions are available

Cycle		Activation	Further information
100	**GENERAL** Definition of generally valid cycle parameters ■ **Q200 SET-UP CLEARANCE** ■ **Q204 2ND SET-UP CLEARANCE** ■ **Q253 F PRE-POSITIONING** ■ **Q208 RETRACTION FEED RATE**	**DEF**-active	Page 1413
105	**DRILLING** Definition of specific drilling cycle parameters ■ **Q256 DIST FOR CHIP BRKNG** ■ **Q210 DWELL TIME AT TOP** ■ **Q211 DWELL TIME AT DEPTH**	**DEF**-active	Page 1414
110	**POCKET MILLING** Definition of specific pocket-milling cycle parameters ■ **Q370 TOOL PATH OVERLAP** ■ **Q351 CLIMB OR UP-CUT** ■ **Q366 PLUNGE**	**DEF**-active	Page 1415
111	**CONTOUR MILLING** Definition of specific contour-milling cycle parameters ■ **Q2 TOOL PATH OVERLAP** ■ **Q6 SET-UP CLEARANCE** ■ **Q7 CLEARANCE HEIGHT** ■ **Q9 ROTATIONAL DIRECTION**	**DEF**-active	Page 1416
125	**POSITIONING** Definition of the positioning behavior with **CYCL CALL PAT** ■ **Q345 SELECT POS. HEIGHT**	**DEF**-active	Page 1416
120	**PROBING** Definition of specific touch probe cycle parameters ■ **Q320 SET-UP CLEARANCE** ■ **Q260 CLEARANCE HEIGHT** ■ **Q301 MOVE TO CLEARANCE**	**DEF**-active	Page 1417

24.5.2 Entering GLOBAL DEF definitions

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **GLOBAL DEF**
- Select the desired **GLOBAL DEF** function, e.g. **100 GENERAL**
- Enter the required definitions

24.5.3 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:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select and define **GLOBAL DEF**
- Select **Insert NC function** again
- Select the desired cycle, e.g. **200 DRILLING**
- > If the cycle includes global cycle parameters, the control superimposes the selection possibility **PREDEF** in the action bar or in the form as a selection menu.

PREDEF

- Select **PREDEF**
- > The control then 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. Simulate your program before executing it
- If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

24.5.4 Global data valid everywhere

These parameters are valid for all **2xx** machining cycles as well as for Cycles **880**, **1017**, **1018**, **1021**, **1022**, **1025** and touch probe cycles **451**, **452**, **453**

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
```

24.5.5 Global data for drilling operations

The parameters apply to the drilling, tapping, and thread milling cycles **200** to **209**, **240**, **241**, **262** to **267**.

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
```

24.5.6 Global data for milling operations with pocket cycles

The parameters apply to the cycles **208**, **232**, **233**, **251** to **258**, **262** to **264**, **267**, **272**, **273**, **275**, and **277**

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
```

24.5.7 Global data for milling operations with contour cycles

The parameters apply to the cycles **20**, **24**, **25**, **27** to **29**, **39**, and **276**

Help graphic	Parameter
	Q2 Path overlap factor? **Q2** x tool radius = stepover factor k Input: **0.0001...1.9999**
	Q6 Set-up clearance? Distance between tool tip and the top surface of the workpiece. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q7 Clearance height? Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q9 Direction of rotation? cw = -1 Machining direction for pockets ■ **Q9** = –1 up-cut milling for pocket and island ■ **Q9** = +1 climb milling for pocket and island Input: **-1**, **0**, **+1**

Example

```
11 GLOBAL DEF 111 CONTOUR MILLING ~
    Q2=+1                   ;TOOL PATH OVERLAP ~
    Q6=+2                   ;SET-UP CLEARANCE ~
    Q7=+50                  ;CLEARANCE HEIGHT ~
    Q9=+1                   ;ROTATIONAL DIRECTION
```

24.5.8 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
```

24.5.9 Global data for probing functions

These parameters are valid for all touch probe cycles **4xx** and **14xx** as well as for Cycles **271**, **286**, **287**, **880**, **1021**, **1022**, **1025**, **1271**, **1272**, **1273**, **1278**

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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at 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
```

24.6 Table access with SQL statements

24.6.1 Fundamentals

Application

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.

Related topics

- Opening, reading and writing to freely definable tables

  Further information: "NC functions for freely definable tables", Page 1394

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function

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.

The SQL commands take effect in the **Program Run** operating mode and the **MDI** application.

Example of transaction:

- Assign Q parameters to table columns for read or write access using **SQL BIND**
- Select data using **SQL EXECUTE** with the **SELECT** instruction
- 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.

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** shows the query result that 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**.

Overview of SQL commands

The control provides the following SQL commands:

Syntax	Function	Further information
SQL BIND	**SQL BIND** creates or disconnects a binding between table columns and Q or QS parameters	Page 1420
SQL SELECT	**SQL SELECT** reads out a single value from a table and does not open any transaction	Page 1421
SQL EXECUTE	**SQL EXECUTE** opens a transaction for selected table columns and table rows or enables the use of other SQL instructions (miscellaneous functions).	Page 1423
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	Page 1427
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	Page 1428
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	Page 1430
SQL UPDATE	**SQL UPDATE** expands the transaction to include the change of an existing row	Page 1431
SQL INSERT	**SQL INSERT** creates a new table row	Page 1433

Notes

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

- 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.

24.6.2 Binding a variable to a table column with SQL BIND

Application

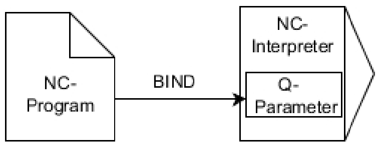
SQL BIND links a Q parameter to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) during data transfer between the **result set** and the NC program.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function



Program any number of bindings with **SQL BIND...**, before using the **FETCH**, **UPDATE**, or **INSERT** commands.

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.

Input

11 SQL BIND Q881 "Tab_example.Position_Nr"	; Bind **Q881** to the "Position_No" column of the "Tab_Example" table

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL BIND	Syntax initiator for the **BIND** SQL command
Q/QL/QR, **QS** or **Q REF**	Variable to be bound
" " or **QS**	Table name and table column, separated by **.** or QS parameter with definition

Notes

- Enter the path of the table or a synonym as the table name.

  Further information: "Executing SQL statements with SQL EXECUTE", Page 1423
- 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.

24.6.3 Reading out a table value with SQL SELECT

Application

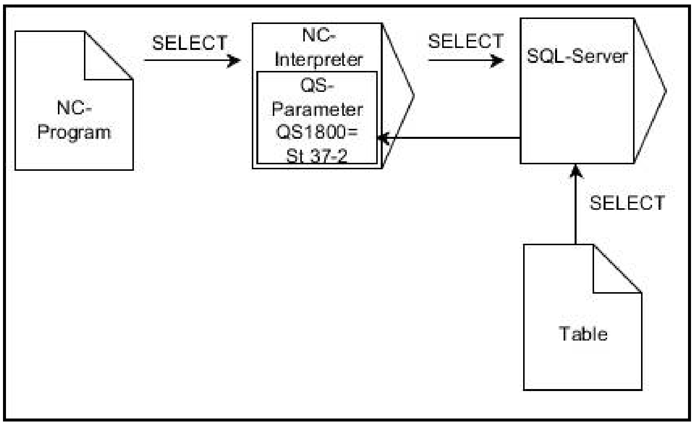
SQL SELECT reads a single value from a table and saves the result in the defined Q parameter.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function



Black arrows and associated syntax show internal processes of **SQL SELECT**

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.

Input

11 SQL SELECT Q5 "SELECT Mess_X FROM Tab_Example WHERE Position_NR==3"	; Save the value of the "Position_No" column of the "Tab_Example" table in **Q5**

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL BIND	Syntax initiator for the **SELECT** SQL command
Q/QL/QR, **QS** or **Q REF**	Variable in which the control stores the result
" " or **QS**	SQL statement or QS parameter with the definition containing: ■ **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)

Notes

- You can select multiple values or multiple columns using the SQL command **SQL EXECUTE** and the **SELECT** statement.
- For the instructions within the SQL command, you can likewise use single or combined QS parameters.

  Further information: "Concatenation of alphanumeric values", Page 1405
- 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.

  Further information: "QPARA tab", Page 180

Example

The result of the following NC programs is 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 value
*** - ...**	

*** - ...**	
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	
*** - ...**	

24.6.4 Executing SQL statements with SQL EXECUTE

Application

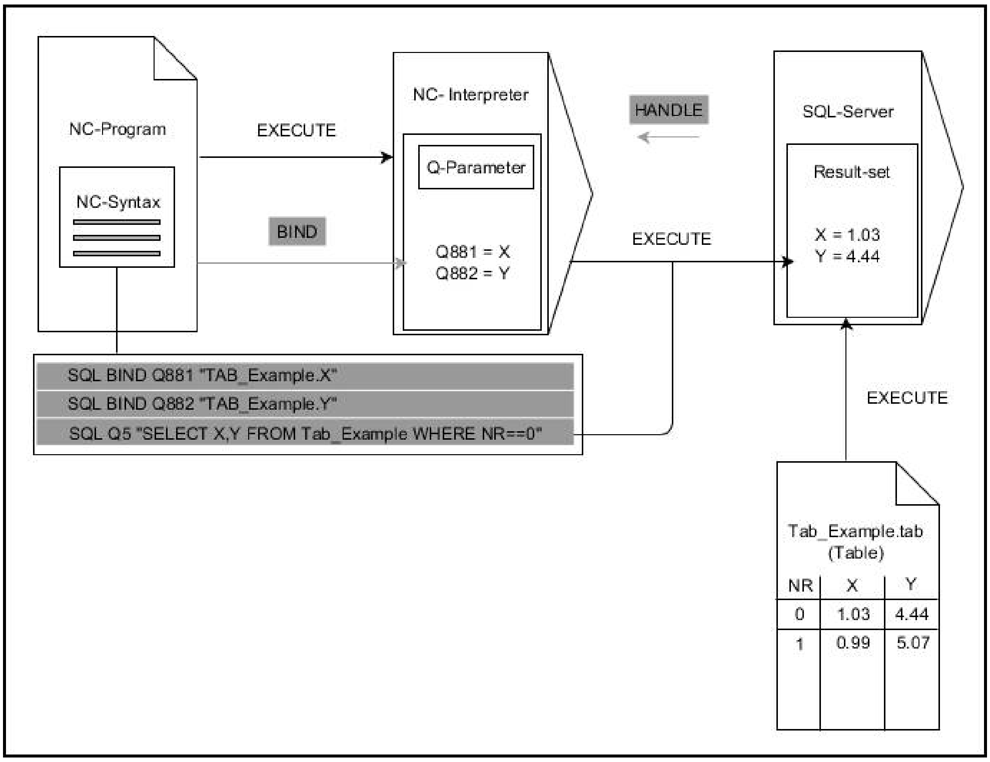
SQL EXECUTE can be used in conjunction with various SQL instructions.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function



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

The control provides the following SQL statements in the **SQL EXECUTE** command:

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate a table
COPY TABLE	Copy table
RENAME TABLE	Rename table
DROP TABLE	Delete a 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

SQL EXECUTE with the SQL instruction SELECT

The SQL server places the data in the **result set** row-by-row. The rows are numbered in ascending order, starting 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.

In the function **SQL ... “SELECT...WHERE..."**, you can enter the search criteria. 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.

In the function **SQL ... “SELECT...ORDER BY..."**, you can enter the ordering criterion. This entry consists of the column designation and the keyword **ASC** for ascending or **DESC** for descending order. If you do not use this option, then rows will be stored in a random order.

With the function **SQL ... "SELECT...FOR UPDATE"**, you can lock the selected rows for other applications. Other applications can continue to read these rows but are unable to change them. If you make changes to the table entries, then it is absolutely necessary to use this option.

Empty result set: If no rows meet the search criterion, then the SQL server returns a valid **HANDLE** without table entries.

Conditions for WHERE entries

Condition	Programming
Equals	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Empty	IS NULL
Not empty	IS NOT NULL
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

Notes

- You can also define synonyms for tables that have not yet been generated.
- The sequence of the columns in the created file corresponds to the sequence within the **AS SELECT** instruction.
- For the instructions within the SQL command, you can likewise use single or combined QS parameters.

  Further information: "Concatenation of alphanumeric values", Page 1405
- 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.

  Further information: "QPARA tab", Page 180

Example

Example: selection of 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: Select 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: Select 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: Define 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"	

0 BEGIN PGM SQL_CREATE_TAB MM	
1 SQL Q10 "CREATE SYNONYM NEW FOR 'TNC:\table\NewTab.TAB'"	; Create synonym
2 SQL Q10 "CREATE TABLE NEW AS SELECT X,Y,Z FROM 'TNC:\prototype_for_NewTab.tab'"	; Create table
3 END PGM SQL_CREATE_TAB MM	

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	

24.6.5 Reading a line from a result set with SQL FETCH

Application

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**.

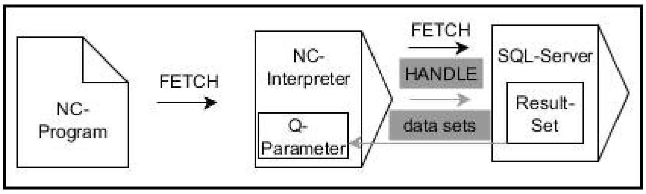
SQL FETCH takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function



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

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL FETCH Q1 HANDLE Q5 INDEX 5 IGNORE UNBOUND UNDEFINE MISSING	; Read out result of transaction **Q5** line 5

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL FETCH	Syntax initiator for the **FETCH** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction
INDEX	Row number within the **result set** as a number or variable If not specified, the control accesses line 0. Optional syntax element
IGNORE UNBOUND	For the machine manufacturer only Optional syntax element
UNDEFINE MISSING	For the machine manufacturer only Optional syntax element

Example

Transfer line 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"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
   Tab_Example"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

24.6.6 Discarding changes to a transaction using SQL ROLLBACK

Application

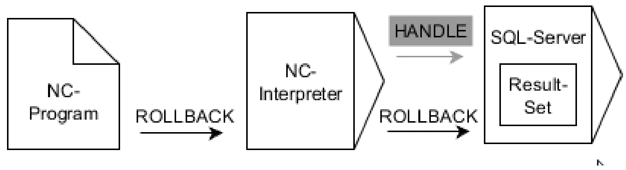
SQL ROLLBACK discards all of the changes and additions of a transaction. The transaction is defined via the **HANDLE** to be specified.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function



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

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

Input

11 SQL ROLLBACK Q1 HANDLE Q5 INDEX 5	; Delete all rows of transaction **Q5** except row 5

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL ROLLBACK	Syntax initiator for the **ROLLBACK** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction
INDEX	Row number within the **Result set** as a number or variable that is retained If not specified, the control discards all changes and additions to the transaction Optional syntax element

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"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
* - ...
41 SQL ROLLBACK Q1 HANDLE Q5

24.6.7 Completing a transaction with SQL COMMIT

Application

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.

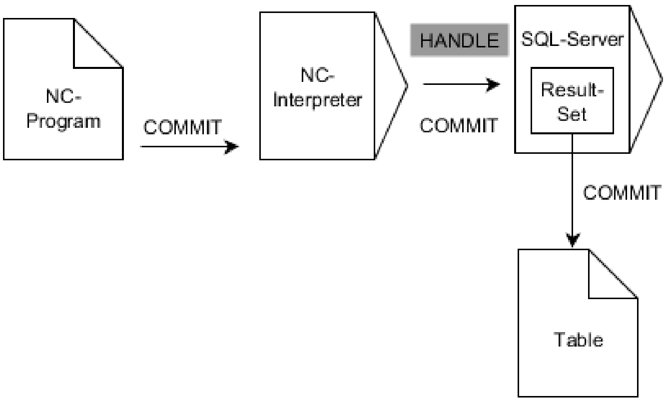
Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function

The assigned **HANDLE** (operation) loses its validity.



Black arrows and associated syntax indicate internal processes of **SQL COMMIT**.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL COMMIT Q1 HANDLE Q5	; Complete all rows of transaction **Q5** and update table

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL COMMIT	Syntax initiator for the **COMMIT** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of 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"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
   Tab_Example"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
* - ...
41 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
* - ...
51 SQL COMMIT Q1 HANDLE Q5
```

24.6.8 Changing the row of a result set with SQL UPDATE

Application

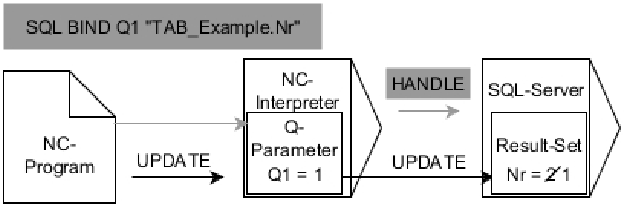
SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**. The control completely overwrites the already existing rows in the **result set**.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function



Black arrows and the associated syntax show internal **SQL UPDATE** processes. Gray arrows and the associated syntax are not directly associated with the **SQL UPDATE** command.

SQL UPDATE takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL UPDATE Q1 HANDLE Q5 index5 RESET UNBOUND	; Complete all rows of transaction **Q5** and update table

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL UPDATE	Syntax initiator for the **UPDATE** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction
INDEX	Row number within the **Result set** as a number or variable If not specified, the control accesses line 0. Optional syntax element
RESET UNBOUND	For the machine manufacturer only Optional syntax element

Note

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 line 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"
* - ...
21 SQL Q5 "SELECT Position_NR,Measure_X,Measure_Y,Measure_Z FROM
   TAB_EXAMPLE"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Program the row number directly

```
31 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

24.6.9 Creating a new row in the result set with SQL INSERT

Application

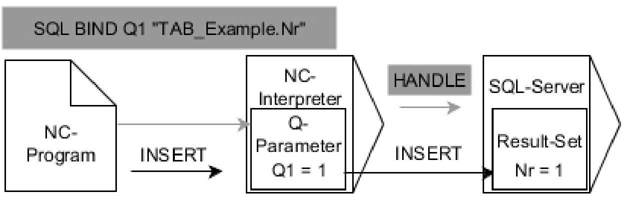
SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  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.

Description of function



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

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.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

```
11 SQL INSERT Q1 HANDLE Q5        ; Create a new row in transaction Q5
```

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL INSERT	Syntax initiator for the **INSERT** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction

Note

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

```
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"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
   Tab_Example"
* - ...
31SQL INSERT Q1 HANDLE Q5
```

24.6.10 Example

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.

Use 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 NR==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.

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	

25

Graphical Programming

25.1 Fundamentals

Application

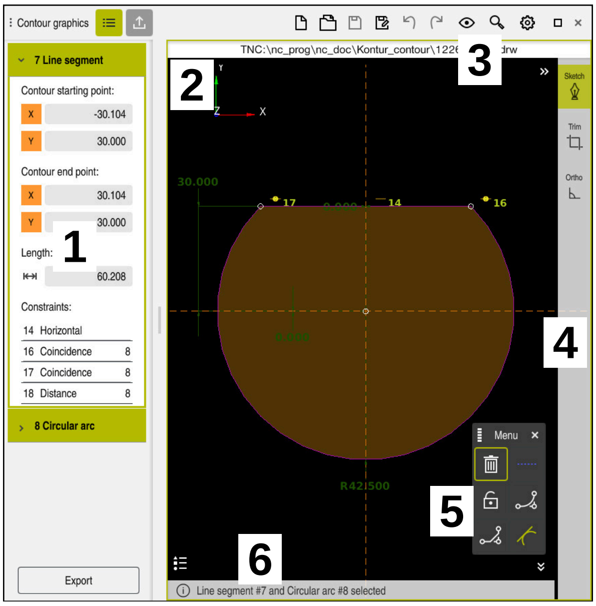
Graphical programming is an alternative to conventional Klartext programming. You draw lines and arcs to create a 2D sketch and then generate a Klartext contour from it. In addition, you can import existing contours from an NC program into the **Contour graphics** workspace and edit them graphically.

You can use graphical programming independently via a separate tab or in the separate **Contour graphics** workspace. If you use graphical programming on its own tab, you cannot open any other workspaces in the **Editor** operating mode on this tab.

Description of function

The **Contour graphics** workspace is available in the **Editor** operating mode.

Screen layout



Screen layout of the **Contour graphics** workspace

The **Contour graphics** workspace contains the following areas:

1 Element information area
2 Drawing area
3 Title bar
4 Toolbar
5 Drawing functions
6 Information bar

Controls and gestures in graphical programming

In graphical programming, you can create a 2D sketch using various elements.

Further information: "First steps in graphical programming", Page 1452

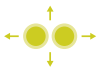
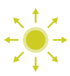
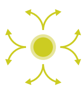
The following elements are available in graphical programming:

- Line segment
- Arc
- Construction point
- Construction line
- Construction circle
- Chamfer
- Rounding arc

Gestures

In addition to the gestures specifically available for graphical programming, you can also use various general gestures in graphical programming.

Further information: "Common gestures for the touchscreen", Page 116

Symbol	Gesture	Meaning
	Tap	Select a point or element
	Long press	Insert construction point
	Two-finger drag	Move the drawing view
	Draw straight elements	Insert **Line segment** element
	Draw circular elements	Insert **Circular arc** element

Icons of the title bar

Besides icons solely available for graphical programming, the title bar of the **Contour graphics** workspace also includes general icons of the control interface.

Further information: "Icons on the control's user interface", Page 123

The control shows the following icons in the title bar:

Icon or shortcut	Meaning
CTRL+O	**Open file**
	View settings
	Show dimensions
	Show constraints
	Show reference axes
	Preset views menu
	Include defined drawing area With this function, the control shows the defined size of the drawing area. You can define the size of the drawing area in the contour settings. **Further information:** "Contour settings window", Page 1444
	Include selected element
	Include drawn elements in drawing area
	Open the **Contour settings** window **Further information:** "Contour settings window", Page 1444

Possible colors

The control shows the elements in the following colors:

Icon	Meaning
	Element A drawn element that is not fully dimensioned is shown by the control in orange as a solid line.
	Construction element Drawn elements can be converted to construction elements. You can use construction elements to obtain additional points for creating your sketch. Construction elements are shown by the control in blue as a dashed line.
	Reference axis The reference axes shown form a Cartesian coordinate system. Dimensioning in graphical programming starts from the intersection of the reference axes. The intersection of the reference axes corresponds to the workpiece preset when exporting the contour data. The control shows reference axes as brown dashed lines.
	Locked element Locked elements cannot be edited. If you want to edit a locked element, you must unlock it first. Locked elements are shown by the control as red solid lines.
	Fully dimensioned element The control shows fully dimensioned elements in dark green. You cannot attach any additional constraints or dimensions to a fully dimensioned element, otherwise the element will be over-determined.
	Contour element The control shows the contour elements between the **Start Point** and **End Point** in the **Export** menu as green solid elements.

Icons in the drawing area

The control shows the following icons in the drawing area:

Icon or shortcut	Designation	Meaning
	Milling direction	The selected **Milling direction** determines whether the defined contour elements are output clockwise or counterclockwise.
	Delete	Deletes all selected elements
	Change the annotation	Switches the display between length and angle dimensions.
	Toggle construction element	This function converts an element into a construction element. Construction elements cannot also be output when exporting a contour.
	Lock element	If this icon is displayed, the selected element is locked against editing. Select the icon to unlock the element.
	Unlock element	If this icon is displayed, the selected element is not locked against editing. Select the icon to lock the element.
	Set the datum	This function moves the selected point to the origin of the coordinate system. All other drawn elements are also moved according to the given distances and dimensions. If necessary, the **Set the datum** function recalculates the existing constraints.
	Corner rounding	Inserts a rounding arc When you select the area of a closed contour, you can round all corners of the contour.
	Chamfer	Inserts a chamfer When you select the area of a closed contour, you can chamfer all corners of the contour.
	Coincidence	This function sets the **Coincidence** constraint for two marked points. When you use this function, the selected points of two elements are connected together. "Coincidence" is used here to refer to these points coinciding.
	Vertical	This function sets the **Vertical** constraint for the selected **Line segment** element. Vertical elements are automatically vertical.
	Horizontal	This function sets the **Horizontal** constraint for the selected **Line segment** element. Horizontal elements are automatically horizontal.
	Perpendicular	This function sets the **Perpendicular** constraint for two selected elements of the type **Line segment**. There is an angle of 90° between perpendicular elements.

Icon or shortcut	Designation	Meaning
	Parallel	This function sets the **Parallel** constraint for two selected elements of the type **Line segment**. When you apply this function, the angle of two lines is aligned. First, the control checks whether there are constraints such as **Horizontal**. Behavior in the case of constraints: ■ If there is a constraint, the **Line segment** without constraint is aligned with the **Line segment** with constraint. ■ If both lines have constraints, the function cannot be applied. The dimension is over-determined. ■ If there are no constraints, the order of selection is decisive. The **Line segment** selected in the second instance is aligned with the **Line segment** first selected.
=	**Equal**	This function sets the **Equal** constraint for two marked elements. When you apply this function, the sizes of two elements are matched (e.g., in length or diameter). First, the control checks whether there are constraints, such as a defined length. Behavior in the case of constraints: ■ If there is a constraint, the element without constraint is aligned with the element with constraint. ■ If both elements have corresponding constraints, the function cannot be applied. The dimension is over-determined. ■ If there are no constraints, the control calculates the average value from the given dimensions.
	Tangential	This function sets the **Tangential** constraint for two marked elements of the **Line segment** and **Circular arc** or **Circular arc** and **Circular arc** type. When you use this function, both arcs and lines are moved. The affected elements come into contact at exactly one point after they are moved and form a tangential transition.
	Symmetry	This function sets the **Symmetry** constraint for a marked element of the type **Line segment** and two marked points of other construction elements. When you apply this function, the control positions the distance of the two points symmetrically to the selected line. If you subsequently change the distance of one of the points, the other point automatically adjusts to the change.
	Point on element	This function sets the **Point on element** constraint for a selected element and a point of another selected element. When you apply this function, the selected point is moved to the selected element.
	Legend	Use this function to show or hide the legend explaining all the controls.
CTRL+D	**Sketch**	To prevent you from unintentionally drawing elements while moving the drawing, you can deactivate drawing mode. Drawing mode remains disabled until you activate it again. If you deactivate drawing mode, the control changes the button to green.

Icon or shortcut	Designation	Meaning
 CTRL+T	**Trim**	If multiple elements overlap, you can use **Trim** mode to shorten elements to the next adjacent element. **Trim** mode remains active until you deactivate it again. If the function is active, the control changes the button to green.
	**Ortho**	With this function, you can only draw rectangular lines. The control does not allow oblique lines or arcs. If the function is active, the control changes the button to green.
CTRL+A	**Select all**	The **Select All** function allows you to mark all drawn elements at once.

Contour settings window

The **Contour settings** window contains the following areas:

- **General information**
- **Sketching**
- **Export**

General information area

The **General information** area contains the following settings:

Setting	Meaning
Plane	You select the plane in which you want to draw by selecting an axis combination. Available planes: ■ **XY** ■ **ZX** ■ **YZ**
Diameter programming	You use a toggle switch to select whether drawn rotation contours in the XZ and YZ planes are interpreted as radius or diameter dimensions during export.
Sketching area width	Default width of the drawing area
Sketching area height	Default height of the drawing area
Decimal places	Number of decimal places for dimensioning

Sketching area

The **Sketching** area contains the following settings:

Setting	Meaning
Rounding radius	Default size for an inserted rounding radius
Chamfer length	Default size for an inserted chamfer
Snap circle size	Size of the snap circle when selecting the elements

Export area

The **Export** area contains the following settings:

Setting	Meaning
Type of circle	You select whether arcs are output as **CC** and **C** or **CR**.
Export as RND	You use a toggle switchto select whether roundings drawn with the **RND** function are also exported as **RND** to the NC program.
CHF output	You use a toggle switch to select whether chamfers drawn with the **CHF** function are also exported as **CHF** to the NC program.

25.1.1 Creating a new contour

To create a new contour:

- Select the **Editor** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **New contour**
- > The control opens the contour in a new tab.

25.1.2 Locking and unlocking elements

If you want to protect an element from editing, you can lock the element. A locked element cannot be edited. If you want to edit the locked element, you must first unlock the element.

To lock and unlock elements in graphical programming:

- Select the drawn element
- Select the **Lock element** function
- > The control locks the element.
- > The control displays the locked element in red.
- Select the **Unlock element** function
- > The control unlocks the element.
- > The control displays the unlocked element in yellow.

Notes

- Set the **Contour settings** before drawing.

  Further information: "Contour settings window", Page 1444
- Dimension each element immediately after drawing. If you do not dimension until the entire contour has been drawn, the contour may move unintentionally.
- You can assign constraints to the drawn elements. To avoid unnecessarily complicating the design, work only with necessary constraints.

  Further information: "Icons in the drawing area", Page 1442
- If you select elements of the contour, the control turns the elements in the menu bar green.

Definitions

File type	Definition
H	NC program in Klartext format
TNCDRW	HEIDENHAIN contour file

25.2 Importing contours into graphical programming

Application

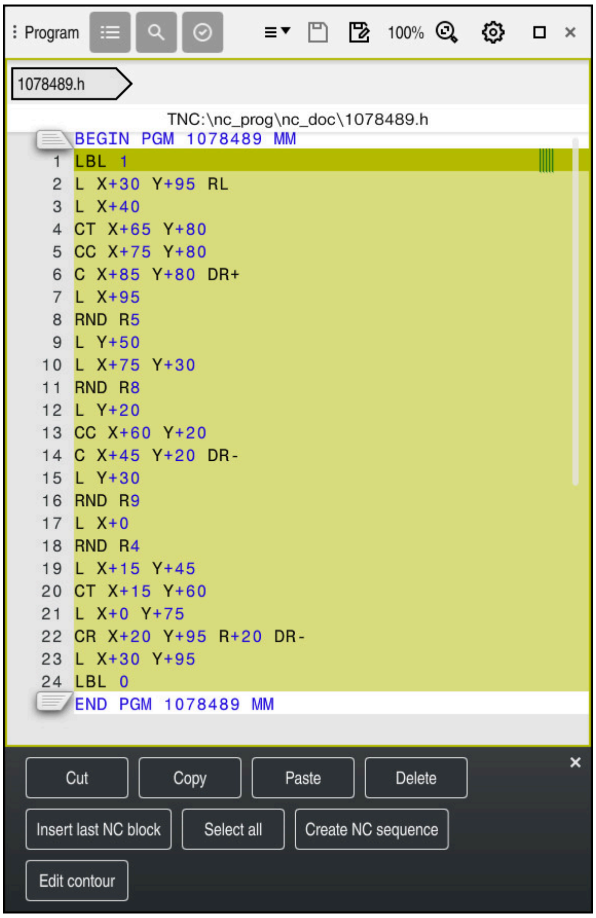
In the **Contour graphics** workspace, you can not only create new contours, but also import contours from existing NC programs and, if necessary, edit them graphically.

Requirements

- Max. 200 NC blocks
- No cycles
- No approach and retraction movements
- No straight lines **LN** (option 9)
- No technology data (e.g., feed rates or additional functions)
- No axis motions that are outside the specified plane (e.g., XY plane)

If you try to import a prohibited NC block into graphical programming, the control will issue an error message.

Description of function



Contour to be imported from the NC program

In graphical programming, all contours consist exclusively of linear or circular elements with absolute Cartesian coordinates.

The control converts the following path functions when importing the contour to the **Contour graphics** workspace:

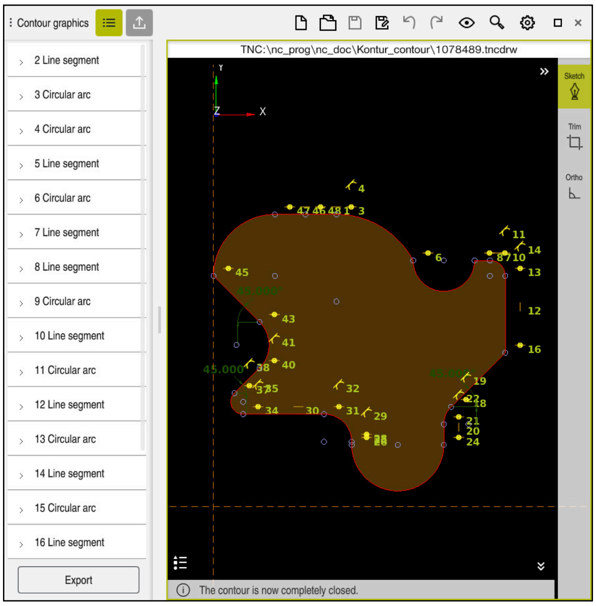
- Circular contour **CT**

  Further information: "Circular path CT", Page 339
- NC blocks with polar coordinates

  Further information: "Polar coordinates", Page 318
- NC blocks with incremental inputs

  Further information: "Incremental entries", Page 321
- Free contour programming **FK**

25.2.1 Importing contours



Imported contour

To import contours from NC programs:

- Select the **Editor** operating mode
- Open an existing NC program with a contour included
- Search for the contour in the NC program
- Hold the first NC block of the contour
- \> The control opens the context menu.
- Select **Mark**
- \> The control shows two marker arrows.
- Select the desired area with the marker arrows
- Select **Edit contour**
- \> The control opens the marked contour area in the **Contour graphics** workspace.

> You can also import contours by dragging the selected NC blocks into the open **Contour graphics** workspace. For this purpose, the control shows a green icon at the right margin of the first highlighted NC block.
>
> **Further information:** "Common gestures for the touchscreen", Page 116

Notes

- In the **Contour settings** window, you can specify whether the dimensions of rotational contours in the XZ plane or YZ plane are interpreted as radius or diameter dimensions.

  Further information: "Contour settings window", Page 1444
- When importing a contour into graphical programming using the **Edit contour** function, all elements are initially locked. Before you begin editing the elements, you must unlock the elements.

  Further information: "Locking and unlocking elements", Page 1445
- You can edit contours graphically and export them after importing.

  Further information: "First steps in graphical programming", Page 1452

  Further information: "Exporting contours from graphical programming", Page 1449

25.3 Exporting contours from graphical programming

Application

The **Export** column in the **Contour graphics** workspace allows you to export newly created or graphically edited contours.

Related topics

- Importing contours

  Further information: "Importing contours into graphical programming", Page 1446
- First steps in graphical programming

  Further information: "First steps in graphical programming", Page 1452

Description of function

The **Export** column provides the following functions:

- **Contour starting point**

  Use this function to define the **Contour starting point**. You can either set the **Contour starting point** graphically or enter an axis value. If you enter an axis value, the control automatically determines the second axis value.
- **Contour end point**

  Use this function to define the **Contour end point**. You can set the **Contour end point** in the same way as the **Contour starting point**.
- **Invert direction**

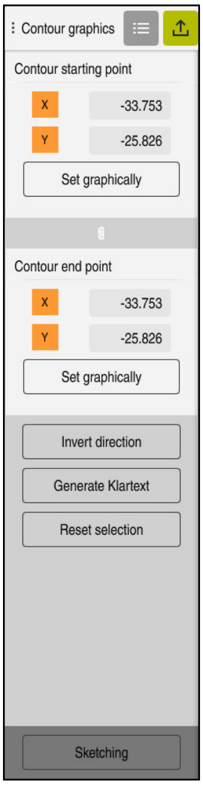
  Use this function to change the programming direction of the contour.
- **Generate Klartext**

  Use this function to export the contour as an NC program or subprogram. The control can only export certain path functions. All generated contours contain absolute Cartesian coordinates.

  Further information: "Contour settings window", Page 1444

  The contour editor can generate the following path functions:
  - Line **L**
  - Circle center **CC**
  - Circular contour **C**
  - Circular contour **CR**
  - Radius **RND**
  - Chamfer **CHF**
- **Reset selection**

  Use this function to deselect a contour.

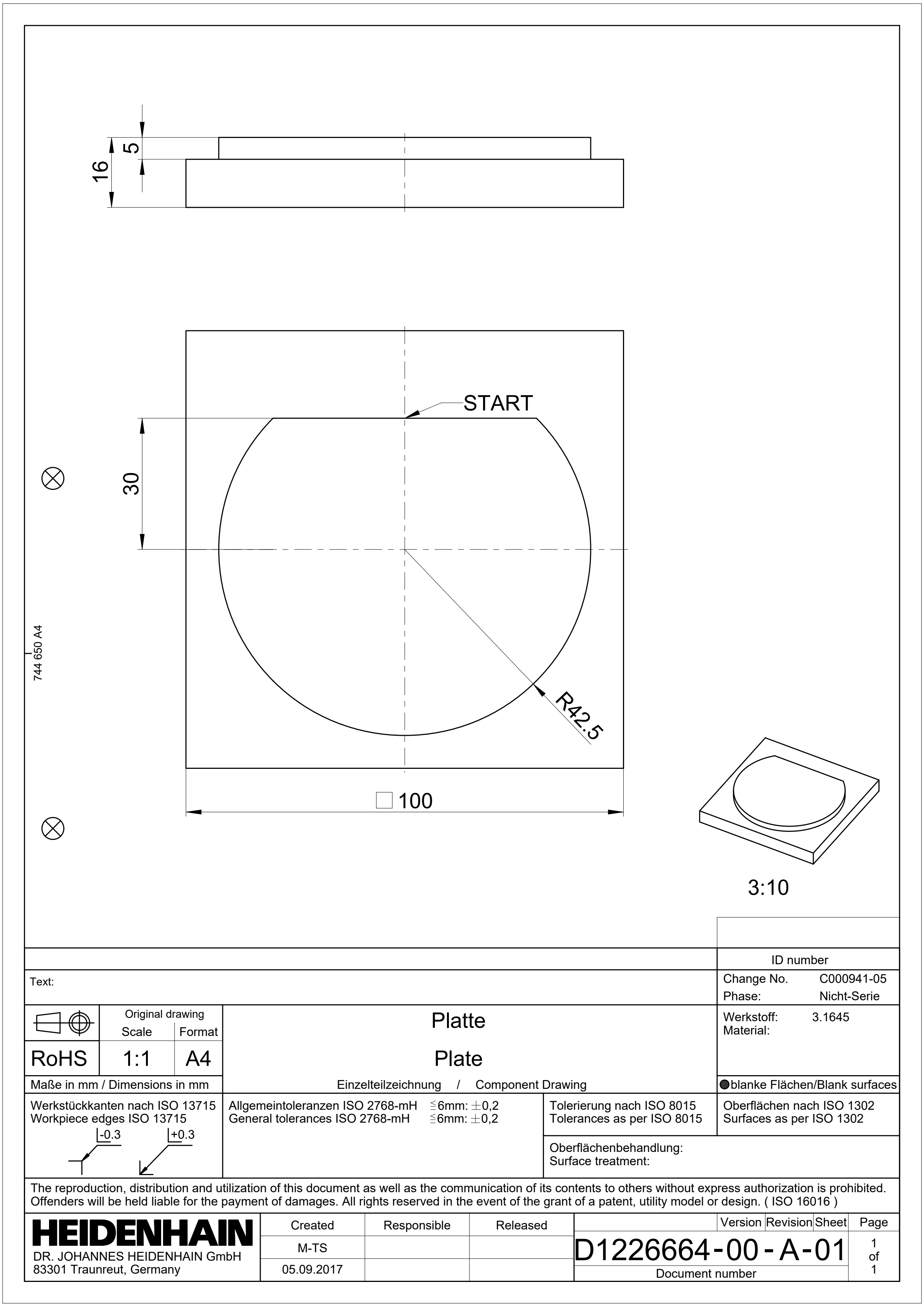


Notes

- You can also use the **Contour starting point** and **Contour end point** functions can to pick up parts of the drawn elements and generate a contour from them.
- You can save drawn contours with the file type ***.tncdrw** to the control.

25.4 First steps in graphical programming

25.4.1 Example task D1226664



25.4.2 Drawing a sample contour

To draw the displayed contour:

- Create a new contour

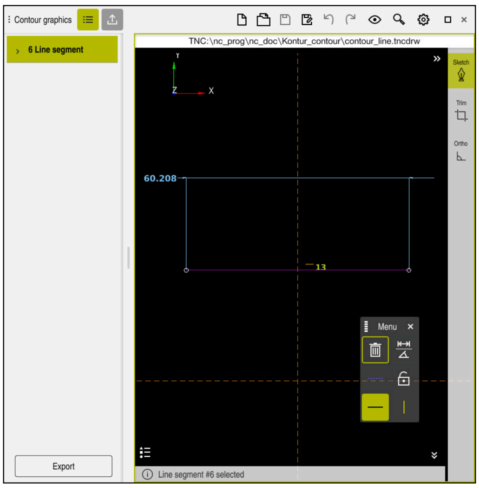
  Further information: "Creating a new contour", Page 1445

- Configure **Contour settings**

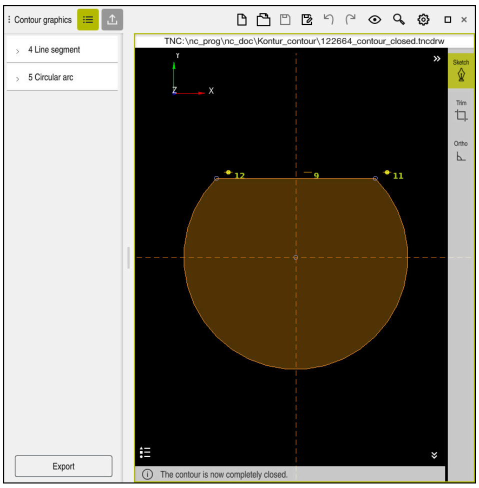
> In the **Contour settings** window, you can define basic settings for drawing. For this example, you can use the standard settings.

Further information: "Contour settings window", Page 1444

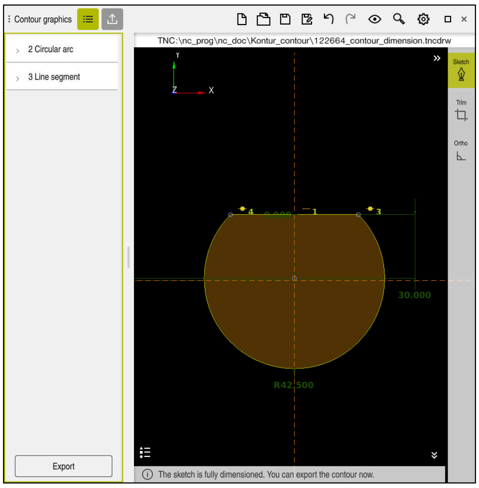
- Draw horizontal **Line segment**
- Select the end point of the drawn line
- > The control shows the X and Y distance of the line to the center.
- Enter Y distance to center (e.g., **30**)
- > The control positions the line according to the condition set.
- Draw **Circular arc** from one end point of the line to the other end point
- > The control displays the closed contour in yellow.
- Select the center point of the arc
- > The control shows the center point coordinates of the arc in **X** and **Y**.
- Enter **0** for the X and Y center point coordinates of the arc
- > The control moves the contour.
- Select drawn arc
- > The control shows the current radius value of the arc.
- Enter radius **42.5**
- > The control adjusts the radius of the arc.
- > The contour is fully defined.



Line drawn



Closed contour



Dimensioned contour

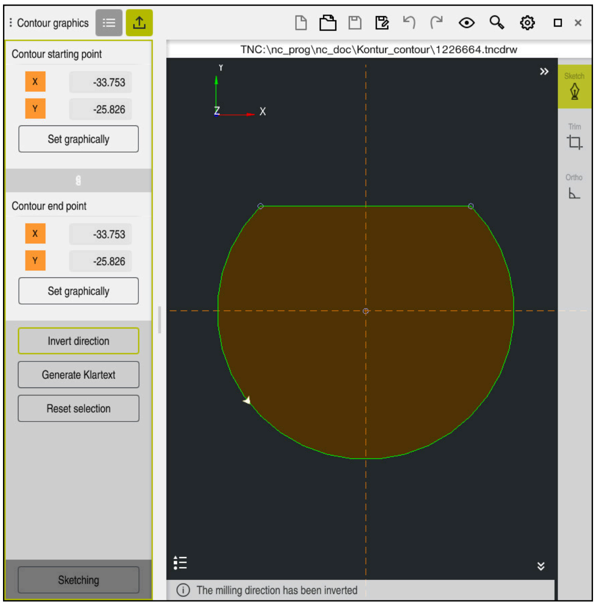
25.4.3 Exporting a drawn contour

To export the drawn contour:

- ▶ Draw contour
- ▶ Select the **Export** column
- > The control displays the **Export** column.
- ▶ Select **Set graphically** in the **Contour starting point** area
- ▶ Select the starting point on the drawn contour
- > The control shows the coordinates of the selected start point, the selected contour and the programming direction.

> You can adjust the programming direction of the contour with the **Invert direction** function.

- ▶ Select the **Generate Klartext** function
- > The control generates the contour based on the defined data.



Selected contour elements in the **Export** column with defined **Milling direction**

26

Opening CAD Files with the CAD-Viewer

26.1 Fundamentals

Application

CAD-Viewer allows you to open the following standardized file types directly on 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
STL	*.stl	■ Binary ■ ASCII

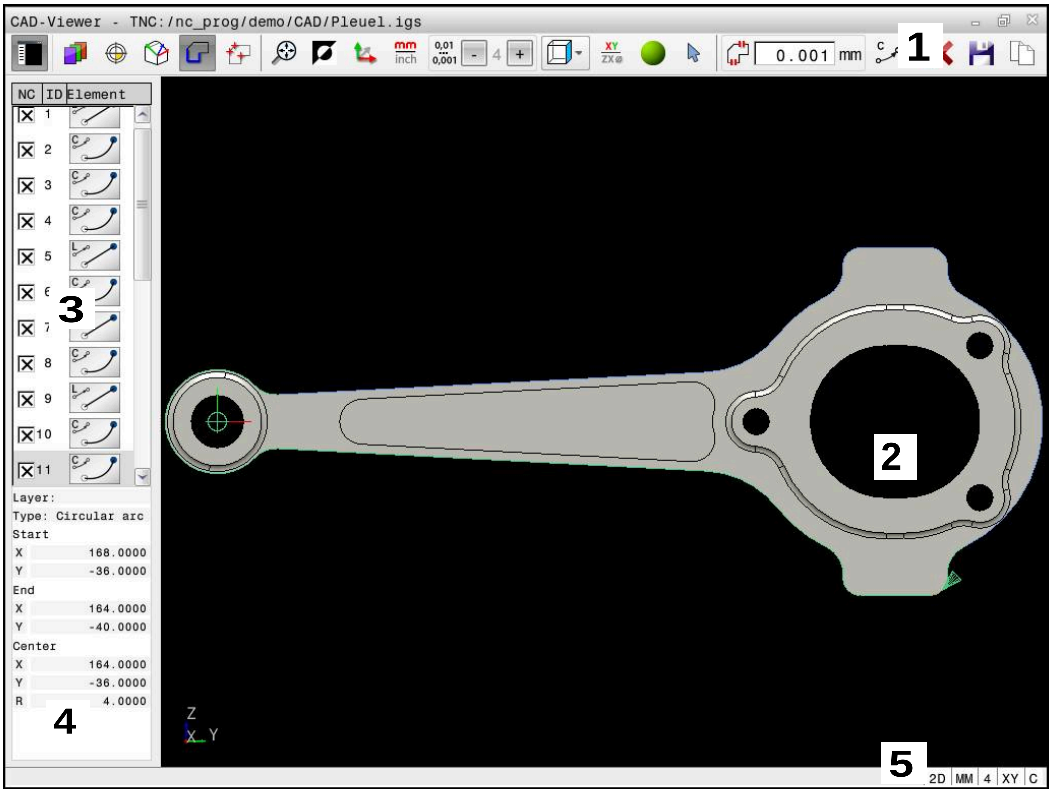
The **CAD-Viewer** runs as a separate application on the third desktop of the control.

Related topics

- Creating 2D sketches on the control
  Further information: "Graphical Programming", Page 1437

Description of function

Screen layout



CAD file open in **CAD-Viewer**

CAD Viewer consists of the following areas:

1 Menu bar

Further information: "Menu bar icons", Page 1458

2 Graphics window

The CAD model is displayed in the graphics window.

3 List View window

The List View window displays information on the active function (e.g., available layers or the position of the workpiece preset).

4 Element Information window

Further information: "Element Information window", Page 1460

5 Status bar

The status bar contains the active settings.

Menu bar icons

The menu bar contains the following icons:

Icon	Function
	Show sidebar Show, enlarge, or hide the List view window
	Display the layer Display the layer(s) in the List View window **Further information:** "Layer", Page 1460
	Preset Define the workpiece preset Workpiece preset has been defined Delete the defined workpiece preset **Further information:** "Workpiece preset in the CAD model", Page 1461
	Datum Set the datum Datum has been set **Further information:** "Workpiece datum in the CAD model", Page 1464
	Contour Select contour (option 42) **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1466
	Positions Select drilling positions (option 42) **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1466
	3D mesh Create a 3D mesh (option 152) **Further information:** "Generating STL files with 3D mesh (option 152)", Page 1472
	Show all Set the zoom to the largest possible view of the complete graphics
	Inverted colors Change the background color (black or white)
	Toggle between 2D and 3D modes
mm inch	Set mm or inches as the unit of measure Internally, **CAD-Viewer** always uses mm for its calculations. If you select inches as the unit of measure, **CAD-Viewer** will convert all values to inches. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1466

Icon	Function
0,01 ... 0,001	**Number of decimal places** Select the resolution. The resolution defines the number of decimal places and the number of positions for linearization. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1466 Default setting: 4 decimal places with **mm**, and 5 decimal places with **inch** as the unit of measure
	Set perspective Switch between various views of the model (e.g., **Top**)
XY	**Axes** Select the working plane: ■ **XY** ■ **YZ** ■ **ZX** ■ **ZXØ** In the **ZXØ** working plane, you can select turning contours (option 50). If you take over a contour or position, the control outputs the NC program in the selected working plane. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1466
	Toggle a 3D model between a solid model and a wire-frame model.
	"Select, add, or remove contour elements" mode The icon shows the current mode. Clicking the icon activates the next mode. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1466
	Undo
	Delete entire list
	Save entire list content to a file
	Copy entire list contents to clipboard The control maintains the contents of the clipboard only as long as **CAD-Viewer** is open.

Element Information window

In the Element Information window, the following information is displayed for the selected element of the CAD file:

- Associated layer
- Element type
- Point type:
  - Point coordinates
- Line type:
  - Coordinates of the starting point
  - Coordinates of the end point
- Circular arc or circle type:
  - Coordinates of the starting point
  - Coordinates of the end point
  - Coordinates of the center point
  - Radius

The control always shows the **X**, **Y**, and **Z** coordinates. In 2D mode, the Z coordinate is dimmed.

Layer

CAD files usually contain multiple layers. The designer uses these layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, hatching, and texts.

The CAD file to be processed must contain at least one layer. The control automatically moves all elements not assigned to a layer to the "anonymous" layer.

If the name of the layer is not shown completely in the window, you can use the **Show sidebar** icon to enlarge this window.

With the **Display the layer** icon, the control shows all the layers of the file in the List view window. Use the checkbox in front of the name to show and hide the individual layers.

When you open a CAD file in **CAD-Viewer**, all available layers are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Notes

- The control does not support binary DXF format. Save the DXF file in ASCII format in the CAD or drawing program.
- Before loading the file into the control, ensure that the name of the file contains only permitted characters.

  Further information: "Permitted characters", Page 1148
- When you select a layer in the list view window, you can use the space bar to show and hide the layer.
- **CAD-Viewer** allows you to open CAD models consisting of any number of triangles.

26.2 Workpiece preset in the CAD model

Application

The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

Related topics

- Presets in the machine

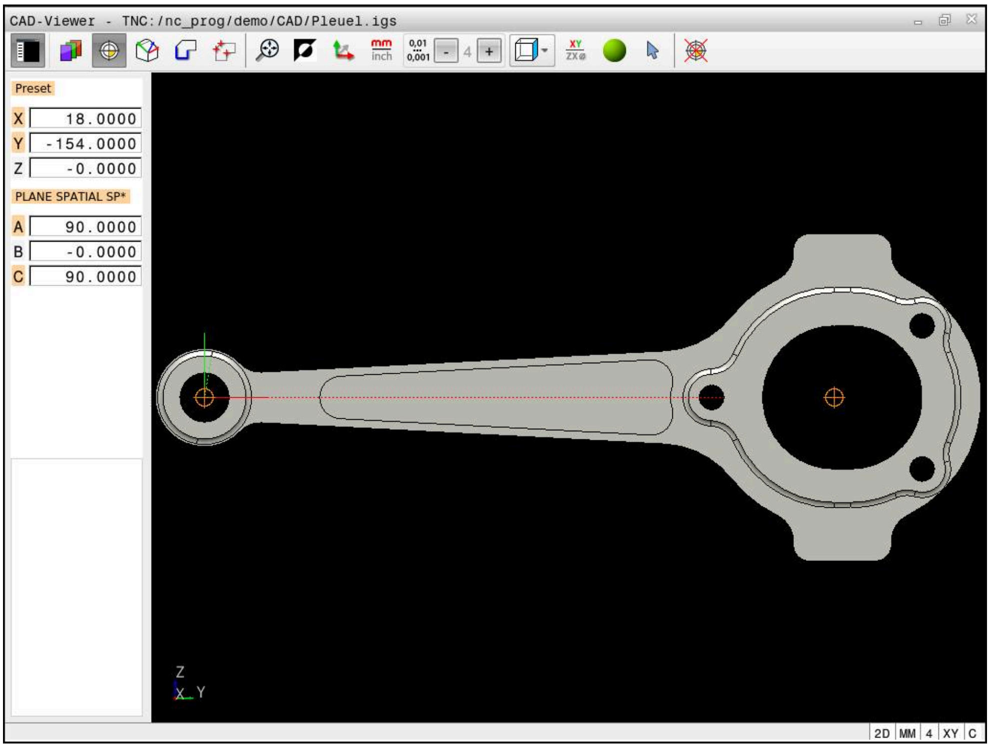
  Further information: "Presets in the machine", Page 210

Description of function

When you select the **Preset** icon, the control displays the following information in the list view window:

- Distance between the defined preset and the drawing datum
- Orientation of the coordinate system with respect to the drawing

The control displays values not equal to 0 in orange.



Workpiece preset in the CAD model

You can position the preset at the following locations:

- By direct input of numerical values into the List View window
- For straight lines:
  - Starting point
  - Midpoint
  - End point
- For circular arcs:
  - Starting point
  - Center point
  - End point
- For full circles:
  - At the quadrant transitions
  - At the center
- At the intersection between:
  - Two straight lines, even if the point of intersection is actually on the extension of one of the lines
  - Straight line and circular arc
  - Straight line and full circle
  - Two circles (regardless of whether a circular arc or a full circle)

If you have set a workpiece preset, the control displays the **Preset** icon in the menu bar with a yellow quadrant.

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;orgin = X... Y... Z...
5 ;orgin_plane_spatial = SPA... SPB... SPC...
```

You can save the information on the workpiece preset and workpiece datum to a file or to the clipboard without having to resort to CAD Import (software option 42).

> The control maintains the contents of the clipboard only as long as **CAD-Viewer** is open.

You can change the preset even after you have selected the contour. The control does not calculate the actual contour data until you save the selected contour in a contour program.

26.2.1 Setting the workpiece preset or workpiece datum and aligning the coordinate system

> - The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
>   **Further information:** "Common gestures for the touchscreen", Page 116
> - The following contents also apply to the workpiece datum. In this case, start by selecting the **Datum** icon.

Setting the workpiece preset or workpiece datum on an individual element

To set the workpiece preset on an individual element:

- Select **Preset**
- Position the cursor on the desired element
- > If you are using a mouse, the control for the element displays selectable presets using gray icons.
- Click the icon at the desired position
- > The control sets the workpiece preset to the selected position. The control turns the icon green.
- Align the coordinate system, if required

Setting the workpiece preset or workpiece datum at the intersection of two elements

You can set the workpiece preset at intersection points of straight lines, full circles and arcs.

To set the workpiece preset at the intersection of two elements:

- Select **Preset**
- Click on the first element
- > The control highlights the element in color.
- Click on the second element
- > The control sets the workpiece preset at the intersection of the two elements. The control marks the workpiece preset with a green icon.
- Align the coordinate system, if required

- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Aligning the coordinate system

The following conditions must be met in order to orient the coordinate system:

- Preset has been defined
- There are elements next to the preset that can be used for the desired orientation

To align the coordinate system:

- Select an element in the positive direction of the X axis
- > The control aligns the X axis.
- > The control changes the angle **C** in the list view window.
- Select an element in the positive direction of the Y axis
- > The control aligns the Y and Z axes.
- > The control changes the angles **A** and **C** in the list view window.

26.3 Workpiece datum in the CAD model

Application

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a tilting operation.

Related topics

- Presets in the machine

  Further information: "Presets in the machine", Page 210

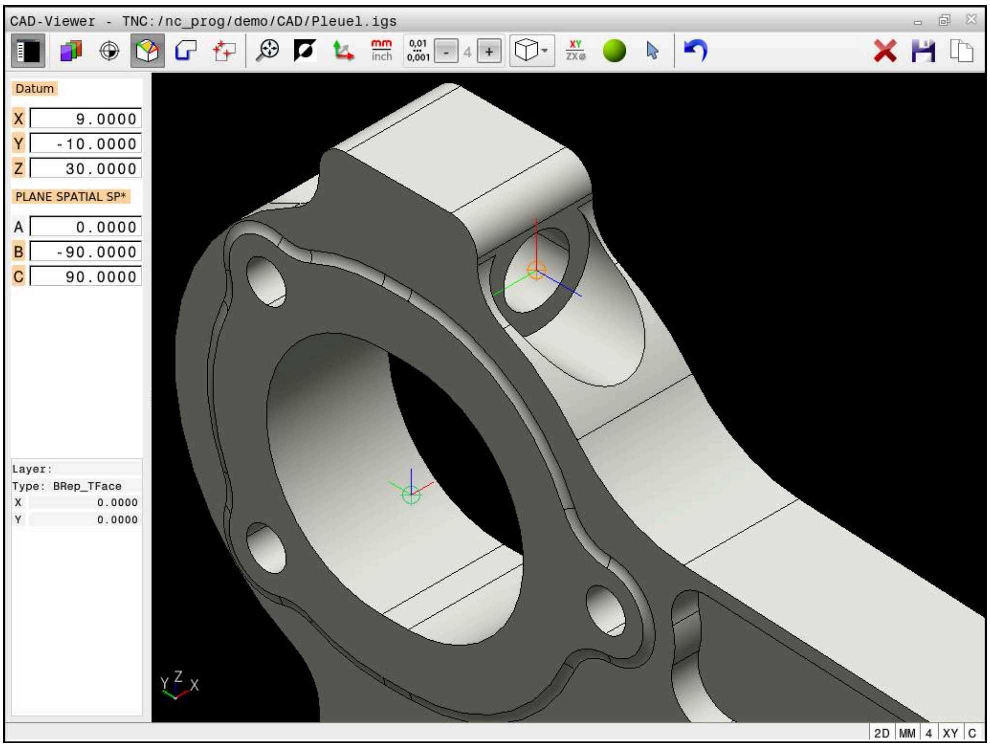
Description of function

When you select the **Datum** icon, the control displays the following information in the list view window:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the coordinate system

You can set a workpiece datum and also move it further by entering values directly in the list view window.

The control displays values not equal to 0 in orange.



Workpiece datum for tilted machining

The datum with the orientation of the coordinate system can be set at the same positions as a preset.

Further information: "Workpiece preset in the CAD model", Page 1461

If you have set a workpiece datum, the control displays the **Datum** icon in the menu bar with a yellow area.

Further information: "Setting the workpiece preset or workpiece datum and aligning the coordinate system", Page 1463

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X... Y... Z...
5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X... Y... Z...
5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

You can save the information on the workpiece preset and workpiece datum to a file or to the clipboard without having to resort to CAD Import (software option 42).

> The control maintains the contents of the clipboard only as long as **CAD-Viewer** is open.

26.4 Applying contours and positions to NC programs with CAD Import (option 42)

Application

You can open CAD files directly on the control to extract contours or machining positions from them. You can then store them as Klartext programs or as point files. Klartext programs acquired in this manner can also be run on older HEIDENHAIN controls, since these contour programs by default contain only **L** and **CC**/**C** blocks.

Related topics

- Using point tables

  Further information: "Point tables", Page 400

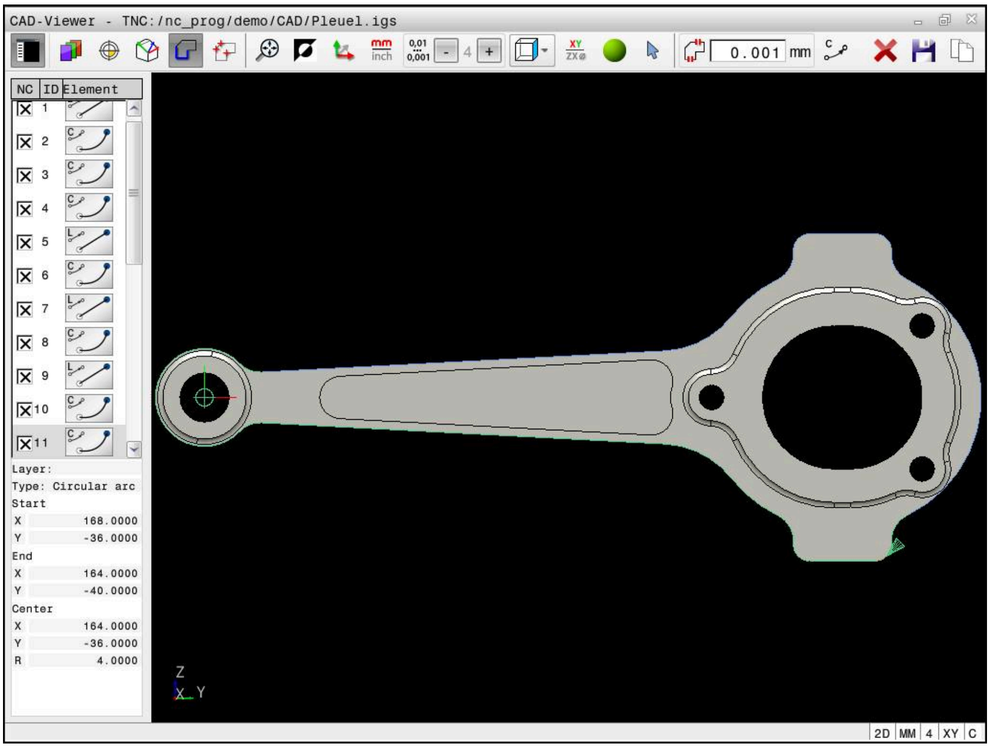
Requirement

- CAD Import (software option 42)

Description of function

To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard. Using the clipboard, you can even transfer the contents to additional software tools (e.g., **Leafpad** or **Gnumeric**).

Further information: "Opening files with additional software", Page 2194



CAD model with marked contour

Icons in the CAD Import

With the CAD Import, the control shows the following additional functions in the menu bar:

Icon	Function
	**Set the transition tolerance** The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred when the drawing was made. The default setting is 0.001 mm
 	**C** or **CR** Arc mode defines whether circular arcs are output in C format or CR format (e.g., for cylinder surface interpolation) in the NC program.
	**Show connections between two positions** Specify whether the control should display the tool path as a dashed line during selection of machining positions
	**Apply path optimization** The control optimizes the tool traverse movement so that there are shorter traverse distances between the machining positions. You reset this optimization by selecting the icon again
	**Find circles according to diameter range. Load center coordinates to the position list** The control opens a pop-up window in which you can filter holes (full circles) based on their size

Applying contours

The following elements can be selected as a contour:

- Line segment
- Circle
- Circular arc
- Polyline
- Any curves (e.g., splines, ellipses)

You can also use the CAD viewer (option 50) to select contours for turning. The icon is grayed out if option 50 is not enabled. Before selecting a turning contour, you must set the preset on the rotary axis. If you select a turning contour, it is saved with Z and X coordinates. In addition, all X coordinate values in turning contours are transferred as diameter values, i.e. the drawing dimensions for the X axis are doubled. All contour elements below the rotary axis cannot be selected and are highlighted gray.

Linearization

During linearization, a contour is divided into individual positions. The CAD Import creates a straight line **L** for each position. With the CAD Import, you can therefore also apply contours that cannot be programmed with the path functions of the control (e.g., splines).

The **CAD-Viewer** linearizes all of the contours that are not in the XY plane. The higher the resolution, the more accurately the control displays the contours.

Applying positions

You can also use the CAD Import to save positions (e.g., for holes).

Three possibilities are available in the pattern generator for defining machining positions:

- Single selection
- Multiple selection within a range
- Multiple selection using search filters

Further information: "Select positions", Page 1470

The following file types are available:

- Point table (**.PNT**)
- Klartext program (**.H**)

If you save the machining positions to a Klartext program, the control creates a separate linear block with a cycle call for every machining position (**L X... Y... Z... F MAX M99**).

> **CAD-Viewer** also recognizes circles that consist of two semi-circles as machining positions.

Multi-selection filter settings

After you have used the quick selection function to mark drilling positions, a pop-up window appears, showing the smallest diameter found to the left and the largest diameter to the right. With the buttons just below the diameter display you can adjust the diameter so that you can transfer the hole diameters that you want.

The following buttons are available:

Icon	Filter setting for the smallest diameter
|<<	Display the smallest diameter found (default setting)
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>	Display the largest diameter found. The control sets the filter for the smallest diameter to the value set for the largest diameter

Icon	Filter setting of largest diameter
<<	Display the smallest diameter found. The control sets the filter for the largest diameter to the value set for the smallest diameter
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>|	Display the largest diameter found (default setting)

26.4.1 Selecting and saving a contour

- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.

  Further information: "Common gestures for the touchscreen", Page 116
- Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.

Selecting a contour with existing contour elements

To select and save a contour with existing contour elements:

- Select **Contour**
- Place the cursor on the first contour element
- > The control shows the suggested direction of rotation as a dashed line.
- If necessary, move the cursor towards the more distant end point.
- > The control changes the suggested direction of rotation.
- Select the contour element
- > The selected contour element is displayed in blue and is marked in the Sidebar window.
- > Other contour elements are shown in green.

The control suggests the contour that deviates least from the suggested direction. To change the suggested contour path, you can select paths independently of the existing contour elements

- Select the last desired contour element
- > All contour elements up to the selected element are shown in blue and are marked in the Sidebar window.

- Select **Save entire list content to a file**
- > The **Define file name for contour program** window opens.
- Enter the desired name
- Select the path to the storage location
- Select **Save**
- > The selected contour is saved as an NC program.

- Alternatively, you can use the **Copy entire list contents to clipboard** icon to copy the selected contour to the clipboard and then paste it into an existing NC program.
- If you select an element with the CTRL key pressed, it is deselected for export.

Selecting paths independent of existing contour elements

To select a path independent of existing contour elements:

- Select **Contour**

- Select **Select**
- > The icon changes, and the control activates the **Add** mode.
- Place the cursor relative to the desired contour element
- > The control displays selectable points:
  - End point or center point of a line or curve
  - Quadrant transitions or center of a circle
  - Points of intersection between existing elements
- Select the desired point
- Select more contour elements

> If the contour element to be extended or shortened is a straight line, the control will extend or shorten the contour element along the same line.
> If the contour element to be extended or shortened is a circular arc, the control will extend or shorten the contour element along the same arc.

Saving the contour as a workpiece blank definition (option 50)

For a workpiece blank definition in turning mode, a closed contour is required.

NOTICE

Danger of collision!

Closed contours must completely lie inside the workpiece blank definition. Otherwise, the system will follow closed contours also along the rotary axis when machining, causing collisions.

- Select or program only those contour elements that are actually required (for example, within the definition of a finished part).

To select a closed contour:

- Select **Contour**
- Select all required contour elements
- Select the starting point of the first element
- > The control closes the contour.

26.4.2 Select positions

> - The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
>   **Further information:** "Common gestures for the touchscreen", Page 116
> - Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.
>   **Further information:** "Selecting and saving a contour", Page 1469

Individual selection

To select individual positions (e.g., holes):

- Select **Positions**
- Position the cursor on the desired element
- > The control shows the circumference and center point of the element in orange.
- Select the desired element
- > The control highlights the selected element in blue and displays it in the list view window.

Multiple selection by area

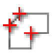
To select multiple positions within an area:

- Select **Positions**

- Select **Select**
- > The icon changes, and the control activates the **Add** mode.
- Drag a box around the area while holding down the left mouse button
- > The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameters found.
- Change the filter settings as needed
- Press **OK**
- > The control highlights all positions of the selected diameter range in blue and shows them in the list view window.
- > The control shows the traversing distance between the positions.

Multiple selection by search filter

To select multiple positions using a search filter:



- Select **Positions**

- **Find circles according to diameter range.** Select **Find circles according to diameter range. Load center coordinates to the position list**
- > The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameter found.

Notes

- Set the correct unit of measure to make sure that **CAD-Viewer** displays correct values.
- Ensure that the unit of measure used in the NC program matches with that used in the **CAD-Viewer**. Elements that have been copied from the **CAD-Viewer** to the clipboard do not contain any information about the unit of measure.
- The control maintains the contents of the clipboard only as long as **CAD-Viewer** is open.
- **CAD-Viewer** also recognizes circles that consist of two semi-circles as machining positions.
- The control also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.

Notes on applying contours

- If you double-click a layer in the list view window, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.
- Select the first contour element such that approach without collision is possible.
- You can even select a contour if the designer has saved it on different layers.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down.
  As an alternative, you can switch to the Remove mode:

26.5 Generating STL files with 3D mesh (option 152)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective files of fixtures and tool holders, for example, or to position STL files generated from the simulation for another machining operation.

Related topics

- Fixture Monitoring (option 40)
  Further information: "Fixture monitoring (option 40)", Page 1171
- Export simulated workpiece as STL file
  Further information: "Exporting a simulated workpiece as STL file", Page 1546
- Using an STL file as workpiece blank
  Further information: "Defining a workpiece blank with BLK FORM", Page 258

Requirement

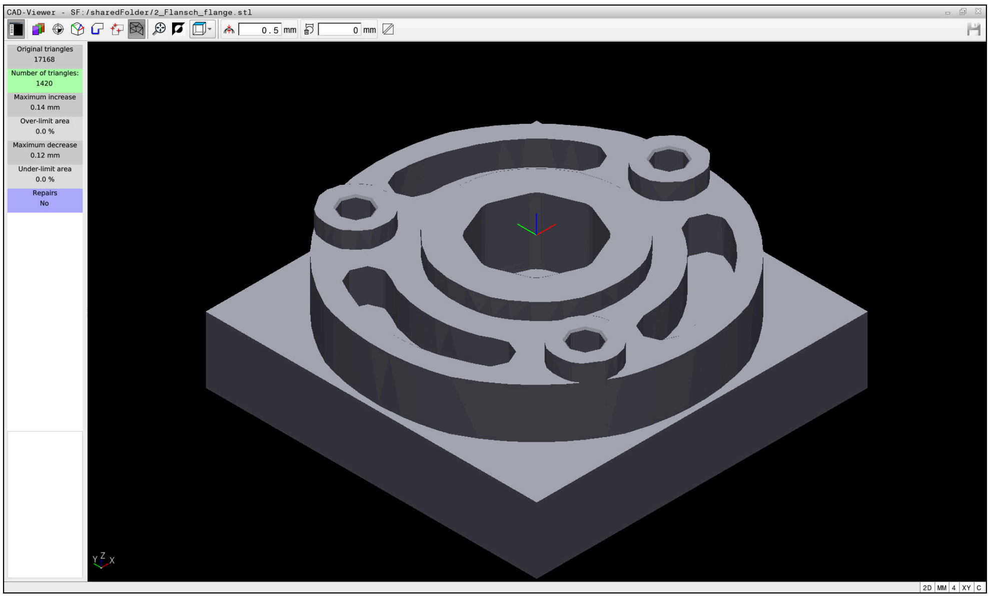
- CAD Model Optimizer (software option 152)

Description of function

When you select the **3D mesh** icon, the control changes to **3D mesh** mode. The control covers the 3D model displayed in **CAD-Viewer** with a mesh of triangles.

The control simplifies the original model and removes errors, such as small holes in a solid or self-intersections of a surface.

You can save the result and use it for various control functions, for example as a workpiece blank with the **BLK FORM FILE** function.



3D model in **3D mesh** mode

The simplified model or parts of it may be smaller or larger than the original model. The result depends on the quality of the original model and the selected settings in **3D mesh** mode.

The Sidebar window shows the following information:

Option	Meaning
Original triangles	Number of triangles in the original model
Number of triangles:	Number of triangles with active settings in the simplified model If this option is highlighted in green, the number of triangles is in the optimum range. You can further reduce the number of triangles using the available functions. **Further information:** "Functions for the simplified model", Page 1474
Maximum increase	Maximum increase of the triangle mesh
Over-limit area	Surface increase in percent compared to the original model
Maximum decrease	Maximum decrease of the triangle mesh compared to the original model
Under-limit area	Surface decrease in percent compared to the original model

Option	Meaning
Repairs	Indicates whether the original model has been repaired or not If it has been repaired, the control indicates the type of repair (e.g., **Hole Int Shells**). This indication consists of the following items: ■ **Hole** **CAD-Viewer** closed holes in the 3D model. ■ **Int** **CAD-Viewer** removed self-intersections. ■ **Shells** **CAD-Viewer** joined multiple separate solids.

In order to use STL files for control functions, the saved files must meet the following requirements:

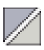
- Max. 20 000 triangles
- Triangular mesh forms a closed shell

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD-Viewer provides the following functions:

Symbol	Function
	**Allowed simplification** Use this function to simplify the output model by the specified tolerance. The higher the value, the more the surfaces may deviate from the original.
	**Remove holes <= diameter** Use this function to remove holes and pockets up to the specified diameter from the original model.
	**Only optimized mesh shown** The control shows the simplified model only.
	**Original is displayed** The control shows the simplified model, superimposed with the original mesh from the original file. You can use this function to evaluate deviations.
	**Save** Use this function to save the simplified 3D model with the selected settings as an STL file.

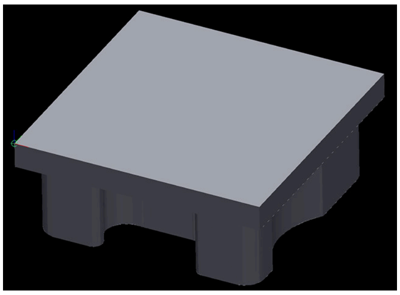
26.5.1 Positioning the 3D model for rear-face machining

To position an STL file for rear-face machining:

- Export the simulated workpiece as an STL file

  Further information: "Saving a simulated workpiece as STL file", Page 1547

- Select the **Files** operating mode
- Select the exported STL file
- > The control opens the CAD files in **CAD-Viewer**.
- Select **Preset**
- > In the Sidebar window, the control displays information on the position of the preset.
- Enter the value of the new preset under **Preset** (e.g., **Z-40**)
- Confirm your input
- Orient the coordinate system by specifying values under **PLANE SPATIAL SP*** (e.g., **A+180** and **C+90**)
- Confirm your input



- Select **3D mesh**
- > The control opens the **3D mesh** mode and simplifies the 3D model using the default settings.
- Further simplify the 3D model using the **3D mesh** mode functions, if required.

  Further information: "Functions for the simplified model", Page 1474

- Select **Save**
- > The control opens the **Define file name for 3D mesh** menu.
- Enter the desired name
- Select **Save**
- > The control saves the STL file positioned for rear-face machining.

> The resulting file can then be used for rear-face machining with the **BLK FORM FILE** function.
>
> **Further information:** "Defining a workpiece blank with BLK FORM", Page 258

27

ISO

27.1 Fundamentals

Application

The ISO 6983 standard defines a universal NC syntax.

Further information: "ISO example", Page 1480

On the TNC7, you can execute and edit NC programs with the supported ISO syntax elements.

Description of function

In connection with ISO programs, the TNC7 provides the following possibilities:

- Transferring files to the control
  Further information: "PC software for data transfer", Page 2191
- Editing ISO programs on the control
  Further information: "ISO syntax", Page 1482
  - In addition to the standardized ISO syntax, you can program HEIDENHAIN-specific cycles as G functions.
    Further information: "Cycles", Page 1501
  - Coding in Klartext syntax allows you to use some NC functions in ISO programs.
    Further information: "Klartext functions in ISO programming", Page 1503
- Testing of NC programs using Simulation mode
  Further information: "Simulation Workspace", Page 1535
- Running NC programs
  Further information: "Program Run", Page 1953

Contents of an ISO program

An ISO program is structured as follows:

ISO syntax	Function
I	File type ISO programs have an ***.i** file name extension.
%NAME G71	Start and end of the program
G71	Unit of measure: mm
G70	Unit of measure: Inch
N10 **N20** **N30** ...	NC block numbers In the optional machine parameter **blockIncrement** (no. 105409), you define the increment between the block numbers.
N99999999	NC block number for the end of the program An NC program is incomplete without this NC block number. The control adds and updates the NC block numbers within the file automatically. The **Program** workspace exclusively shows successive numbers without taking the defined increment into account.
G01 X+0 Y+0 ...	NC functions

Further information: "Contents of an NC program", Page 212

Contents of an NC block

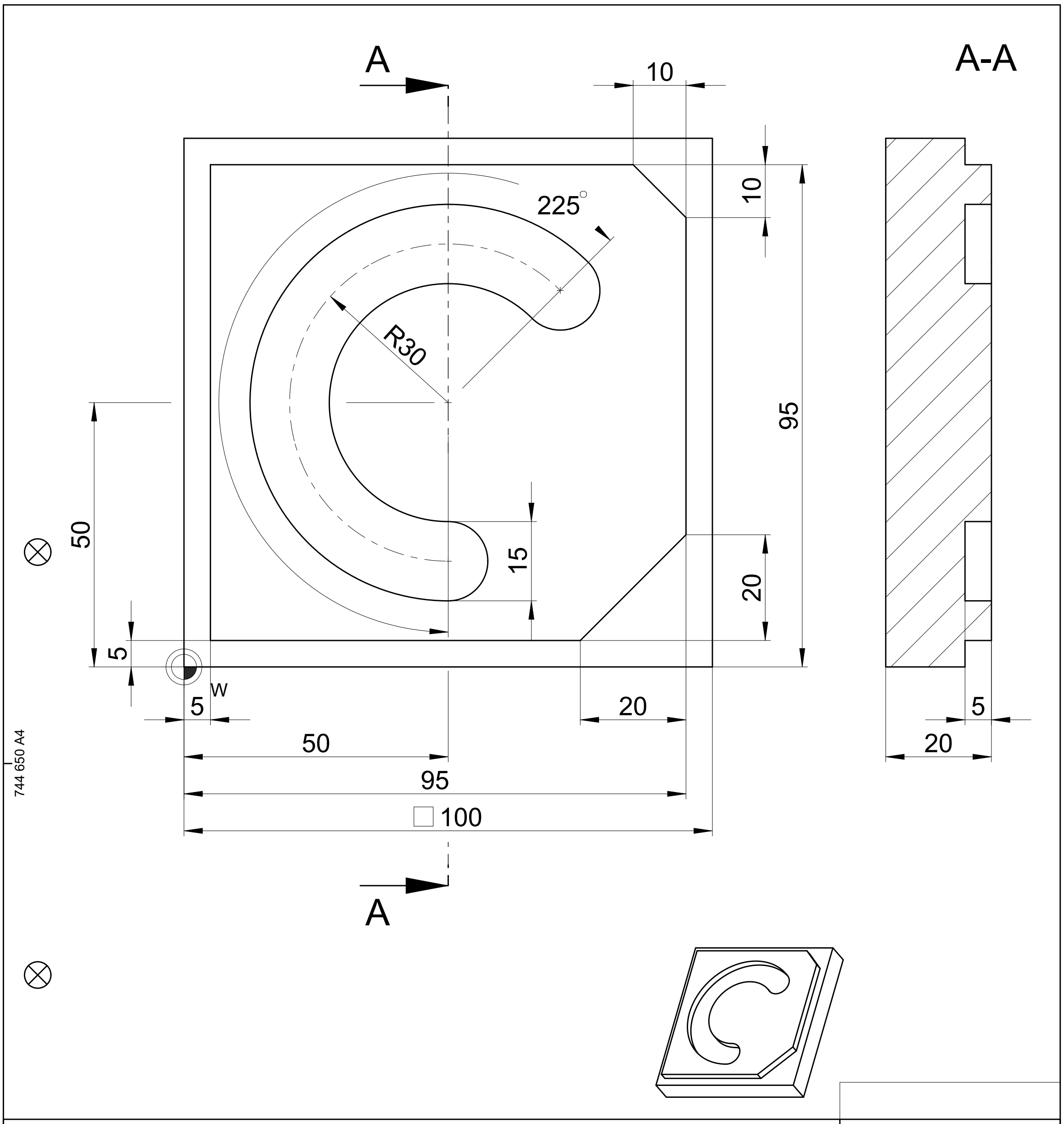
```
N110 G01 G90 X+10 Y+0 G41 F3000 M3
```

An NC block contains the following syntax elements:

ISO syntax	Function
G01	Start of syntax
G90	Absolute or incremental input **Further information:** "Absolute and incremental input", Page 1482
X+10 Y+0	Coordinates **Further information:** "Fundamentals of coordinate definitions", Page 318
G41	Tool radius compensation **Further information:** "Tool radius compensation", Page 1493
F3000	Feed rate **Further information:** "Feed rate", Page 1484
M3	Miscellaneous functions (M functions) **Further information:** "Miscellaneous Functions", Page 1317

ISO example

Example task 1338459



Text:

ID number

Change No. C000941-05

Phase: Nicht-Serie

Original drawing	Scale	Format	Platte / Plate	Werkstoff: Material:
RoHS	1:1	A4	Einzelteilzeichnung / Component Drawing	●blanke Flächen/Blank surfaces

Maße in mm / Dimensions in mm

Werkstückkanten nach ISO 13715
Workpiece edges ISO 13715
-0.3 +0.3

Allgemeintoleranzen ISO 2768-mH ≦6mm: ±0,2
General tolerances ISO 2768-mH ≦6mm: ±0,2

Tolerierung nach ISO 8015
Tolerances as per ISO 8015

Oberflächen nach ISO 1302
Surfaces as per ISO 1302

Oberflächenbehandlung:
Surface treatment:

HEIDENHAIN
DR. JOHANNES HEIDENHAIN GmbH
83301 Traunreut, Germany

Created	Responsible	Released	Version Revision Sheet	Page
M-TS			D1358459-00-A-01	1 of 1
05.08.2021			Document number	

Example solution 1338459

% 1339889 G71	
N10 G30 G17 X+0 Y+0 Z-40	; Workpiece blank definition
N20 G31 X+100 Y+100 Z+0	; Workpiece blank definition
N30 T16 G17 S6500	; Tool call
N40 G00 G90 Z+250 G40 M3	; Clearance height in the tool axis
N50 G00 X-20 Y-20	; Pre-positioning in the machining plane
N60 G00 Z+5	; Pre-positioning in the tool axis
N70 G01 Z-5 F3000 M8	; Feed to working depth
N80 G01 X+5 Y+5 G41 F700	; First contour point
N90 G26 R8	; Approach function
N100 G01 Y+95	; Straight line
N110 G01 X+95	
N120 G24 R10	; Chamfer
N130 G01 Y+5	
N140 G24 R20	
N150 G01 X+5	
N160 G27 R8	; Departure function
N170 G01 X-20 Y-20 G40 F1000	; Clearance height in the machining plane
N180 G00 Z+250	; Clearance height in the tool axis
N190 T6 G17 S6500	; Tool call
N200 G00 G90 Z+250 G40 M3	
N210 G00 X+50 Y+50 M8	
N220 CYCL DEF 254 CIRCULAR SLOT ~	
Q215=+0 ;MACHINING OPERATION ~	
Q219=+15 ;SLOT WIDTH ~	
Q368=+0.1 ;ALLOWANCE FOR SIDE ~	
Q375=+60 ;PITCH CIRCLE DIAMETR ~	
Q367=+0 ;REF. SLOT POSITION ~	
Q216=+50 ;CENTER IN 1ST AXIS ~	
Q217=+50 ;CENTER IN 2ND AXIS ~	
Q376=+45 ;STARTING ANGLE ~	
Q248=+225 ;ANGULAR LENGTH ~	
Q378=+0 ;STEPPING ANGLE ~	
Q377=+1 ;NR OF REPETITIONS ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-5 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	

Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q366=+2 ;PLUNGE ~	
Q385=+500 ;FINISHING FEED RATE ~	
Q439=+0 ;FEED RATE REFERENCE	
N230 G79	; Cycle call
N240 G00 Z+250 M30	
N99999999 % 1339889 G71	

Notes

- ISO programs can be edited using any text editor (e.g., **Leafpad**).
- You can call a Klartext program within an ISO program (e.g., to benefit from the possibilities of graphical programming).

  Further information: "Calling an NC program", Page 1490

  Further information: "Graphical Programming", Page 1437
- You can call a Klartext program within an ISO program (e.g., to use NC functions that are available only for Klartext programming).

  Further information: "Machining with polar kinematics with FUNCTION POLARKIN", Page 1295

27.2 ISO syntax

Absolute and incremental input

The control provides the following possibilities to enter dimensions:

Syntax	Meaning
G90	Absolute input always references an origin. For Cartesian coordinates, the origin is the datum, and for polar coordinates the origin is the pole and the angle reference axis.
G91 corresponds to the **I** Klartext syntax	Incremental input always references the previously programmed coordinates. For Cartesian coordinates, these are the values in the **X**, **Y**, and **Z** axes, and for polar coordinates, the values of the polar coordinate radius **R** and the polar coordinate angle **H**.

Tool axis

In some NC functions, you can select a tool axis in order, for example, to define the machining plane.

> 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 differentiates between the following tool axes:

Syntax	Working plane
G17 corresponds to the **Z** tool axis	**XY**, as well as **UV**, **XV**, **UY**
G18 corresponds to the **Y** tool axis	**ZX**, as well as **VW**, **YW**, **VZ**
G19 corresponds to the **X** tool axis	**YZ**, as well as **WU**, **ZU**, **WX**

Workpiece blank

Use the **G30** and **G31** NC functions to define a cuboid workpiece blank for simulation in the NC program.

You define the cuboid by entering a MIN point for the bottom front left corner and a MAX point for the top rear right corner.

N10 G30 G17 X+0 Y+0 Z-40	Define MIN point
N20 G31 X+100 Y+100 Z+0	Define MAX point

G30 and **G31** correspond to the Klartext syntax **BLK FORM 0.1** and **BLK FORM 0.2**.

Further information: "Defining a workpiece blank with BLK FORM", Page 258

With **G17**, **G18**, and **G19**, you define the tool axis.

Further information: "Tool axis", Page 1483

With the Klartext syntax, you can additionally define the following workpiece blanks:

- Cylindrical workpiece blank with **BLK FORM CYLINDER**

  Further information: "Cylindrical workpiece blank with BLK FORM CYLINDER", Page 262
- Rotationally symmetric workpiece blank with **BLK FORM ROTATION**

  Further information: "Rotationally symmetric workpiece blank with BLK FORM ROTATION", Page 263
- STL file as workpiece blank with **BLK FORM FILE**

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 264

Tools

Tool call

With the **T** NC function, you call a tool in the NC program.

T corresponds to the **TOOL CALL** Klartext syntax.

Further information: "Tool call by TOOL CALL", Page 304

With **G17**, **G18**, and **G19**, you define the tool axis.

Further information: "Tool axis", Page 1483

Cutting data

Spindle speed

The spindle speed **S** is defined as spindle revolutions per minute (rpm).

Alternatively, the constant cutting speed **VC** in meters per minute (m/min) can be defined.

N110 T1 G17 S(VC = 200)	; Tool call with constant cutting speed

Further information: "Spindle speed S", Page 309

Feed rate

The feed rate for linear axes is defined in millimeters per minute (mm/min).

In inch programs, the feed rate must be defined in 1/10 inch/min.

The feed rate for rotary axes is defined in degrees per minute (°/min).

The feed rate can be defined with an accuracy of three decimal places.

Further information: "Feed rate F", Page 310

Tool definition

With the **G99** NC function, you can define the dimensions/allowance of a tool.

> Refer to your machine manual.
> A tool definition created with **G99** is a machine-dependent function.
> HEIDENHAIN recommends using tool management for the definition of tools instead of **G99**!
> **Further information:** "Tool management ", Page 297

110 G99 T3 L+10 R+5	; Define tool

G99 corresponds to the **TOOL DEF** Klartext syntax.

Further information: "Tool pre-selection by TOOL DEF", Page 311

Tool pre-selection

When you use the **G51** NC function, the control prepares a tool in the magazine, thus reducing the tool-change time.

> Refer to your machine manual.
> A tool pre-selection defined with **G99** is a machine-dependent function.

110 G51 T3	; Tool pre-selection

G51 corresponds to the **TOOL DEF** Klartext syntax.

Further information: "Tool pre-selection by TOOL DEF", Page 311

Path functions

Straight line

Cartesian coordinates

With the **G00** and **G01** NC functions, you program a straight movement in rapid traverse or with a machining feed rate in any desired direction.

N110 G00 Z+100 M3	; Straight line at rapid traverse
N120 G01 X+20 Y-15 F200	; Straight line at machining feed rate

If the feed rate was programmed using a numerical value, it is active only up to the NC block in which a new feed rate is programmed. **G00** is active only for the NC block in which it was programmed. When the NC block programmed with **G00** has been executed, the feed rate programmed most recently with a numerical value becomes active again.

> Make sure to program rapid traverse movements exclusively with the **G00** NC function instead of very 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.

G00 and **G01** correspond to the **L** Klartext syntax with **FMAX** and **F**.

Further information: "Straight line L", Page 326

Polar coordinates

With the **G10** and **G11** NC functions, you program a straight movement in rapid traverse or with a machining feed rate in any desired direction.

N110 I+0 J+0	; Pole
N120 G10 R+10 H+10	; Straight line at rapid traverse
N130 G11 R+50 H+50 F200	; Straight line at machining feed rate

The polar coordinate radius **R** corresponds to the **PR** Klartext syntax.

The polar coordinate angle **H** corresponds to the **PA** Klartext syntax.

G10 and **G11** correspond to the **LP** Klartext syntax with **FMAX** and **F**.

Further information: "Straight line LP", Page 346

Chamfer

With the **G24** NC function, you can insert a chamfer between two straight lines. The chamfer size references the point of intersection you are programming using the straight line.

N110 G01 X+40 Y+5	; Straight line at machining feed rate
N120 G24 R12	; Chamfer at machining feed rate
N130 G01 X+5 Y+0	; Straight line at machining feed rate

The value following the **R** syntax element corresponds to the chamfer size.

G24 corresponds to the **CHF** Klartext syntax.

Further information: "Chamfer CHF", Page 328

Rounding arc

With the **G25** NC function, you can insert a rounding arc between two straight lines. The rounding arc references the point of intersection you are programming using the straight line.

N110 G01 X+40 Y+25	; Straight line at machining feed rate
N120 G25 R5	; Rounding arc at machining feed rate
N130 G01 X+10 Y+5	; Straight line at machining feed rate

G25 corresponds to the **RND** Klartext syntax.

The value following the **R** syntax element corresponds to the radius of the rounding arc.

Further information: "Rounding RND", Page 330

Circle center

Cartesian coordinates

With the **I**, **J**, and **K** or **G29** NC functions, you define the circle center.

N110 I+25 J+25	; Circle center in the XY plane

N110 G00 X+25 Y+25	; Pre-positioning on a straight line
N120 G29	; Circle center at the last position

- **I**, **J**, and **K**
  The circle center is defined in this NC block.
- **G29**
  The control assumes the most recently programmed position as the circle center.

I, **J**, and **K** or **G29** correspond to the **CC** Klartext syntax with or without axis values.

Further information: "Circle center point CC", Page 332

With **I** and **J**, you define the circle center in the **X** and **Y** axes. In order to define the **Z** axis, program **K**.

Further information: "Circular path in another plane", Page 343

Polar coordinates

With the **I**, **J**, and **K** or **G29** NC functions, you define a pole. All polar coordinates reference the pole.

N110 I+25 J+25	; Pole

- **I**, **J**, and **K**
  The pole is defined in this NC block.
- **G29**
  The control takes over the most recently programmed position as the pole.

I, **J**, and **K** or **G29** correspond to the **CC** Klartext syntax with or without axis values.

Further information: "Polar coordinate datum at pole CC", Page 345

Circular arc with center

Cartesian coordinates

With the **G02**, **G03**, and **G05** NC functions, you program a circular path around a circle center.

N110 I+25 J+25	; Circle center
N120 G03 X+45 Y+25	; Circular path around circle center

- **G02**

  Circular path in clockwise direction, corresponds to the **C** Klartext syntax with **DR-**.
- **G03**

  Circular path in counterclockwise direction, corresponds to the **C** Klartext syntax with **DR+**.
- **G05**

  Circular path without direction of rotation, corresponds to the **C** Klartext syntax without **DR**.

  The control uses the most recently programmed direction of rotation.

Further information: "Circular path C ", Page 334

Polar coordinates

With the **G12**, **G13**, and **G15** NC functions, you program a circular path around a defined pole.

N110 I+25 J+25	; Pole
N120 G13 H+180	; Circular path around pole

- **G12**

  Circular path in clockwise direction, corresponds to the **CP** Klartext syntax with **DR-**.
- **G13**

  Circular path in counterclockwise direction, corresponds to the **CP** Klartext syntax with **DR+**.
- **G15**

  Circular path without direction of rotation; corresponds to the **CP** Klartext syntax without **DR**.

  The control uses the most recently programmed direction of rotation.

The polar coordinate angle **H** corresponds to the **PA** Klartext syntax.

Further information: "Circular path CP around pole CC", Page 348

Circular path with a defined radius

Cartesian coordinates

With the **G02**, **G03**, and **G05** NC functions, you program a circular path with a defined radius. If you are programming a radius, no circle center is required.

N110 G03 X+70 Y+40 R+20	; Circular path with a defined radius

- **G02**
  Circular path in clockwise direction, corresponds to the **CR** Klartext syntax with **DR-**.
- **G03**
  Circular path in counterclockwise direction, corresponds to the **CR** Klartext syntax with **DR+**.
- **G05**
  Circular path without direction of rotation; corresponds to the **CR** Klartext syntax without **DR**.
  The control uses the most recently programmed direction of rotation.

Further information: "Circular path CR", Page 336

Circular arc with a tangential transition

Cartesian coordinates

With the **G06** NC function, you program a circular path with a tangential transition to the previous path function.

N110 G01 X+25 Y+30 F300	; Straight line
N120 G06 X+45 Y+20	; Circular path with tangential transition

G06 corresponds to the **CT** Klartext syntax.

Further information: "Circular path CT", Page 339

Polar coordinates

With the **G16** NC function, you program a circular path with a tangential transition to the previous path function.

N110 G01 G42 X+0 Y+35 F300	; Straight line
N120 I+40 J+35	; Pole
N130 G16 R+25 H+120	; Circular path with tangential transition

The polar coordinate radius **R** corresponds to the **PR** Klartext syntax.

The polar coordinate angle **H** corresponds to the **PA** Klartext syntax.

G16 corresponds to the **CTP** Klartext syntax.

Further information: "Circular path CTP", Page 350

Contour approach and departure

With the **G26** and **G27** NC functions, you can approach or depart the contour smoothly using a circle segment.

N110 G01 G40 G90 X-30 Y+50	; Starting point
N120 G01 G41 X+0 Y+50 F350	; First contour point
N130 G26 R5	; Tangential approach
*** - ...**	
N210 G27 R5	; Tangential exit
N220 G00 G40 X-30 Y+50	; End point

HEIDENHAIN recommends the use of the more powerful **APPR** and **DEP** NC functions. In some cases, these NC functions combine multiple NC blocks for approaching and departing the contour.

G41 and **G42** correspond to the **RL** and **RR** Klartext syntax.

Further information: "Approach and departure functions with Cartesian coordinates", Page 358

You can also use polar coordinates when programming the **APPR** and **DEP** NC functions.

Further information: "Approach and departure functions with polar coordinates", Page 372

Programming techniques

Subprograms and program-section repeats

Programming techniques are useful in structuring your NC program and avoiding unnecessary repeats. By using subprograms, you need to define machining positions for multiple tools only once, for example. Program-section repeats, on the other hand, help you avoid multiple programming of identical, successive NC blocks or program sequences. By combining and nesting these two programming techniques, you can keep your NC programs rather short and restrict changes to a few central program locations.

Further information: "Subprograms and program section repeats with the label LBL", Page 384

Defining labels

With the **G98** NC function, you define a new label in the NC program.

Each label must be unambiguously identifiable in the NC program by a number or a name. If a number or a name exists twice in an NC program, the control shows a warning ahead of the NC block.

If you define a label after **M30** or **M2**, it corresponds to a subprogram. Subprograms must always be concluded with a **G98 L0**. This number is the only one which may exist any number of times in the NC program.

N110 G98 L1	; Start of subprogram defined by a number
N120 G00 Z+100	, Retract at rapid traverse
N130 G98 L0	; End of subprogram

N110 G98 L "UP"	; Start of subprogram defined by a name

G98 L corresponds to the **LBL** Klartext syntax.

Further information: "Defining a label with LBL SET", Page 384

Calling a subprogram

With the **L** NC function, you call a subprogram programmed after **M30** or **M2**.

When the control reads the **L** NC function, it will jump to the defined label and continue execution of the NC program from this NC block. When the control reads **G98 L0**, it will jump back to the next NC block after the call with **L**.

N110 L1	; Call subprogram

L without **G98** corresponds to the **CALL LBL** Klartext syntax.

Further information: "Calling a label with CALL LBL", Page 385

Program-section repeat

Program-section repeats allow you to have a particular program section executed any number of times. The program section must start with a **G98 L** label definition and end with **L**. With the numeral after the decimal point, you can define optionally how often you want the control to repeat this program section.

N110 L1.2	; Call label 1 twice

L without **98** and the numeral after the decimal point correspond to the **CALL LBL REP** Klartext syntax.

Further information: "Program-section repeats", Page 387

Selection functions

Further information: "Selection functions", Page 388

Calling an NC program

With the **%** NC function, you can call another, separate NC program from within an NC program.

N110 %TNC:\nc_prog\reset.i	; Call NC program

% corresponds to the **CALL PGM** Klartext syntax.

Further information: "Calling an NC program with PGM CALL", Page 388

Activating a datum table in the NC program

With the **%:TAB:** NC function, you can activate a datum table from within an NC program.

N110 %:TAB: "TNC:\table\zeroshift.d"	; Activate datum table

%:TAB corresponds to the **SEL TABLE** Klartext syntax.

Further information: "Activating the datum table in the NC program", Page 1034

Selecting a point table

With the **%:PAT:** NC function, you can activate a point table from within an NC program.

N110 %:PAT: "TNC:\nc_prog \positions.pnt"	; Activate point table

%:PAT corresponds to the **SEL PATTERN** Klartext syntax.

Further information: "Selecting the point table in the NC program with SEL PATTERN", Page 401

Selecting an NC program with contour definitions

With the **%:CNT:** NC function, you can select another NC program with a contour definition from within an NC program.

N110 %:PAT: "TNC:\nc_prog\contour.h"	; Select NC program with contour definition

Further information: "Graphical Programming", Page 1437

%:CNT corresponds to the **SEL CONTOUR** Klartext syntax.

Further information: "Selecting an NC program with contour definition", Page 413

Selecting and calling an NC program

With the **%:PGM:** NC function, you can select another, separate NC program. With the **%<>%** NC function, you call the selected NC program at a different location in the active NC program.

N110 %:PGM: "TNC:\nc_prog\reset.i"	; Select NC program
*** - ...**	
N210 %<>%	; Call the selected NC program

%:PGM: and **%<>%** correspond to the **SEL PGM** and **CALL SELECTED PGM** Klartext syntax.

Further information: "Calling an NC program with PGM CALL", Page 388

Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 390

Defining an NC program as a cycle

With the **G: :** NC function, you can define another NC program as a machining cycle from within an NC program.

N110 G: : "TNC:\nc_prog\cycle.i"	; Define NC program as a machining cycle

G: : corresponds to the **SEL CYCLE** Klartext syntax.

Further information: "Defining and calling an NC program as cycle", Page 483

Cycle call

For cycles that remove material, you have to enter not only the cycle definition, but also the cycle call in the NC program. The call always refers to the fixed cycle that was last defined in the NC program.

The control provides the following options for calling a cycle:

Syntax	Meaning
G79 corresponds to the **CYCLE CALL** Klartext syntax	The control calls the most recently programmed machining cycle at the last programmed position.
G79 PAT corresponds to the **CYCLE CALL PAT** Klartext syntax	The control calls the most recently programmed machining cycle at all positions you have defined in a point table.
G79|G01 corresponds to the **CYCLE CALL POS** Klartext syntax	The control calls the most recently programmed machining cycle at the position you defined in the NC block with **G79|G01**.
M89 and **M99**	With **M99**, the control executes the most recently programmed machining cycle at the most recently programmed position. With **M89**, the control executes the most recently programmed machining cycle after each positioning block until it reads **M99**.

N110 G79 M3	; Call cycle

N110 G79 PAT F200 M3	; Call cycle at all positions in the point table

N110 G79|G01 G90 X+0 X+25	; Call cycle at the defined position

N110 G01 X+0 X+25 M89	; Call cycle at the defined position and for each new positioning block
N120 G01 X+25 Y+25	
N130 G01 X+50 Y+25 M99	; Call cycle for the last time at the defined position

Further information: "Calling cycles", Page 481

Tool radius compensation

When tool radius compensation is active, the control will no longer reference the positions in the NC program to the tool center point, but to the cutting edge.

An NC block can contain the following tool radius compensations:

Syntax	Meaning
G40 corresponds to the **R0** Klartext syntax	Reset an active tool radius compensation, positioning based on the tool center point
G41 corresponds to the **RL** Klartext syntax	Tool radius compensation, on the left of the contour
G42 corresponds to the **RR** Klartext syntax	Tool radius compensation, on the right of the contour

Further information: "Tool radius compensation", Page 1114

Miscellaneous functions (M functions)

Use miscellaneous functions to activate or deactivate functions of the control and to influence the behavior of the control.

Further information: "Miscellaneous Functions", Page 1317

G38 corresponds to the **STOP** Klartext syntax.

Further information: "Miscellaneous functions M and the STOP function ", Page 1318

Programming variables

The control provides the following options for programming variables in ISO programs:

Function group	Further information
Basic arithmetic operations	Page 1495
Trigonometric functions	Page 1496
Circle calculations	Page 1497
Jump commands	Page 1498
Special functions	Page 1500
String functions	Corresponds to the Klartext syntax Page 1400
Counters	Corresponds to the Klartext syntax Page 1409
Calculations using formulas	Corresponds to the Klartext syntax Page 1397
Function for the definition of complex contours	Corresponds to the Klartext syntax Page 410

The control distinguishes between the **Q**, **QL**, **QR**, and **QS** variable types (parameter types).

Further information: "Variable Programming", Page 1361

Not all NC functions for programming variables are available in ISO programs (e.g., accessing tables with SQL statements).

Further information: "Table access with SQL statements", Page 1417

Basic arithmetic operations

With the **D01** through **D05** functions, you can calculate values within your NC program. If you want to calculate with variables, you need to assign an initial value to each variable by means of the **D00** function.

The control provides the following functions:

Syntax	Meaning
D00	Assignment Assign a value or the **Undefined** status
D01	Addition Calculate and assign the sum of two values
D02	Subtraction Calculate and assign the difference of two values.
D03	Multiplication Calculate and assign the product of two values.
D04	Division Calculate and assign the quotient of two values Restriction: You cannot divide by 0
D05	Square root Calculate and assign the square root of a number Restriction: You cannot calculate a square root from a negative value

```
N110 D00 Q5 P01 +60            ; Assignment Q5 = 60
```

```
N110 D01 Q1 P01 -Q2 P02 -5     ; Addition Q1 = −Q2+(−5)
```

```
N110 D02 Q1 P01 +10 P02 +5     ; Subtraction Q1 = +10−(+5)
```

```
N110 D03 Q2 P01 +3 P02 +3      ; Multiplication Q2 = 3*3
```

```
N110 D04 Q4 P01 +8 P02 +Q2     ; Division Q4 = 8/Q2
```

```
N110 D05 Q20 P01 4             ; Square root Q20 =√4
```

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "Basic arithmetic folder", Page 1374

> HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.
>
> **Further information:** "Formulas in the NC program", Page 1397

Trigonometric functions

You can use these functions to calculate trigonometric functions for purposes such as programming variable triangular contours.

The control provides the following functions:

Syntax	Meaning
D06	Sine Calculate and assign the sine of an angle in degrees
D07	Cosine Calculate and assign the cosine of an angle in degrees
D08	Root of the sum of squares Calculate and assign the length based on two values (e.g., to calculate the third side of a triangle).
D13	Angle 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°)

```
N110 D06 Q20 P01 -Q5            ; Sine, Q20 = sin(−Q5)
```

```
N110 D07 Q21 P01 -Q5            ; Cosine, Q21 = cos(−Q5)
```

```
N110 D08 Q10 P01 +5 P02 +4      ; Root of the sum of squares, Q10 = √(5²+4²)
```

```
N110 D13 Q20 P01 +10 P02 -Q1    ; Angle, Q20 = arctan(25/−Q1)
```

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "Trigonometric functions folder", Page 1376

> HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.
>
> **Further information:** "Formulas in the NC program", Page 1397

Circle calculation

These functions allow you to calculate the center of a circle and the radius of the circle based on the coordinates of three or four points on the circle (e.g., the position and size of a circle segment).

The control provides the following functions:

Syntax	Meaning
D23	Circle data from three points on the circle The control saves the determined values in three successive Q parameters so that you only need to program the number of the first variable.
D24	Circle data from four points on the circle The control saves the determined values in three successive Q parameters so that you only need to program the number of the first variable.

```
N110 D23 Q20 P01 Q30    ; Circle data from three points on the circle
```

```
N110 D24 Q20 P01 Q30    ; Circle data from four points on the circle
```

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "Circle calculation folder", Page 1378

Jump commands

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.

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

The control provides the following functions:

Syntax	Meaning
D09	Jump if equal If both values are equal, the control jumps to the defined label.
	Jump if undefined If the variable is undefined, the control jumps to the defined label.
	Jump if defined If the variable is defined, the control jumps to the defined label.
D10	Jump if not equal If both values are not equal, the control jumps to the defined label.
D11	Jump if greater than If the first value is greater than the second one, the control jumps to the defined label.
D12	Jump if less than If the first value is less than the second one, the control jumps to the defined label.

N110 D09 P01 +Q1 P02 +Q3 P03 "LBL"	; Jump if equal
N110 D09 P01 +Q1 IS UNDEFINED P03 "LBL"	; Jump if undefined
N110 D09 P01 +Q1 IS DEFINED P03 "LBL"	; Jump if defined
N110 D10 P01 +10 P02 -Q5 P03 10	; Jump if not equal
N110 D11 P01 +Q1 P02 +10 P03 QS5	; Jump if greater than
N110 D12 P01 +Q5 P02 +0 P03 "LBL"	; Jump if less than

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: " Jump commands folder", Page 1380

Functions for freely definable tables

You can open any free definable table and subsequently write to it or read from it.

The control provides the following functions:

Syntax	Meaning
D26	Open a freely definable table **Further information:** "Opening a freely definable table with FN 26: TABOPEN", Page 1394
D27	Write to a freely definable table **Further information:** "Writing to a freely definable table with FN 27: TABWRITE", Page 1394
D28	Read from a freely definable table **Further information:** "Reading a freely definable table with FN 28: TABREAD", Page 1395

N110 D26 TNC:\DIR1\TAB1.TAB	; Open a freely definable table

N110 Q5 = 3.75	; Define the value for the **Radius** column
N120 Q6 = -5	; Define the value for the **Depth** column
N130 Q7 = 7,5	; Define the value for the **D** column
N140 D27 P01 5/“Radius,Depth,D“ = Q5	; Write defined values to the table

N110 D28 Q10 = 6/“X,Y,D“*	; Read numerical values from the **X**, **Y**, and **D** columns
N120 D28 QS1 = 6/“DOC“*	; Read the alphanumeric value from the **DOC** column

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Special functions

The control provides the following functions:

Syntax	Meaning
D14	Output error messages **Further information:** "Output error messages with FN 14: ERROR", Page 1381 **Further information:** "Preassigned error numbers for FN 14: ERROR", Page 2265
D16	Output formatted texts **Further information:** "Outputting text formatted with FN 16: F-PRINT", Page 1382
D18	Read system data **Further information:** "Read system data with FN 18: SYSREAD", Page 1388 **Further information:** "System data", Page 2271
D19	Transfer values to the PLC **Further information:** "Transfer values to PLC with FN 19: PLC", Page 1389
D20	Synchronize NC and PLC **Further information:** "Synchronizing NC and PLC with FN 20: WAIT FOR", Page 1390
D29	Transfer values to the PLC **Further information:** "Transferring values to PLC with FN 29: PLC", Page 1391
D37	Create user-defined cycles **Further information:** "Creating your own cycles with FN 37: EXPORT", Page 1391
D38	Send information from the NC program **Further information:** "Sending information from the NC program with FN 38: SEND", Page 1392

N110 D14 P01 1000	; Output error message no. 1000
N110 D16 P01 F-PRINT TNC:\mask.a / TNC: \Prot1.txt	; Display the output file with **D16** on the control screen
N110 D18 Q25 ID210 NR4 IDX3	; Save the active dimension factor of the Z axis in **Q25**
N110 D38 /"Q-Parameter Q1: %F Q23: %F" P02 +Q1 P02 +Q23	; Write the values of **Q1** and **Q23** to the log

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

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. The functions **D19**, **D20**, **D29**, and **D37** enable HEIDENHAIN, the machine manufacturer, and suppliers 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 a risk of collision during the execution of these functions 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

27.3 Cycles

Fundamentals

In ISO programs, you can use selected cycles with Klartext syntax in addition to the NC functions with ISO syntax. Programming is identical to Klartext programming.

The numbers of the Klartext cycles correspond to the numbers of the G functions. There are exceptions for earlier cycles that have numbers below **200**. In these cases, the corresponding G function number is mentioned in the cycle description.

Further information: "Machining Cycles", Page 475

The following cycles are not available in ISO programs:

- Cycle **1 POLAR PRESET**
- Cycle **3 MEASURING**
- Cycle **4 MEASURING IN 3-D**
- Cycle **26 AXIS-SPECIFIC SCALING**

HEIDENHAIN recommends using the more powerful **PLANE** functions instead of Cycle **G80 WORKING PLANE**. With the **PLANE** functions, you can choose freely between axis or spatial angles for programming.

Further information: "PLANE SPATIAL", Page 1059

Datum shift

With the **G53** or **G54** NC functions, you can program datum shifts. **G54** shifts the workpiece datum to the coordinates you define directly within this function. **G53** uses coordinate values from a datum table. A datum shift allows machining operations to be repeated at any locations on the workpiece.

N110 G54 X+0 Y+50	; Shift the workpiece datum to the defined coordinates

N110 G53 P01 10	; Shift the workpiece datum to the coordinates of table row 10

To reset a datum shift:

- Define the value **0** for each axis in function **G54**
- In function **G53**, select a table row where all columns have the value **0**

The control displays the following information in the **Status** workspace:

- Name and path of the active datum table
- Active datum number
- Comment from the **DOC** column of the active datum number

Notes

In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.

- Datums from a datum table always reference the current workpiece preset.
- Before shifting the workpiece datum by means of a datum table, you need to activate the datum table with **%:TAB:**

  Further information: "Activating a datum table in the NC program", Page 1490
- If you do not use **%:TAB:**, you have to activate the datum table manually.

  Further information: "Activating the datum table manually", Page 1034

27.4 Klartext functions in ISO programming

Fundamentals

In ISO programs, you can use selected NC functions with Klartext syntax in addition to the NC functions with ISO syntax. Programming is identical to Klartext programming.

For more information about programming, refer to the respective chapters describing the individual NC functions.

The following NC functions are available only in Klartext programs:

- Pattern definitions with **PATTERN DEF**

  Further information: "Pattern definition with PATTERN DEF", Page 419
- NC functions for coordinate transformations: **TRANS DATUM**, **TRANS MIRROR**, **TRANS ROTATION**, and **TRANS SCALE**

  Further information: "NC functions for coordinate transformation", Page 1045
- File functions: **FUNCTION FILE** and **OPEN FILE**

  Further information: "Programmable file functions", Page 1158
- Functions for machining with parallel axes: **PARAXCOMP** and **PARAXMODE**

  Further information: "Working with the parallel axes U, V and W", Page 1284
- Programs that use normal vectors

  Further information: "CAM-generated NC programs", Page 1301
- Table access with SQL statements

  Further information: "Table access with SQL statements", Page 1417

28

User Aids

28.1 Help workspace

Application

In the **Help** workspace the control displays a help graphic for the current syntax element of an NC function or the **TNCguide** integrated product aid.

Related topics

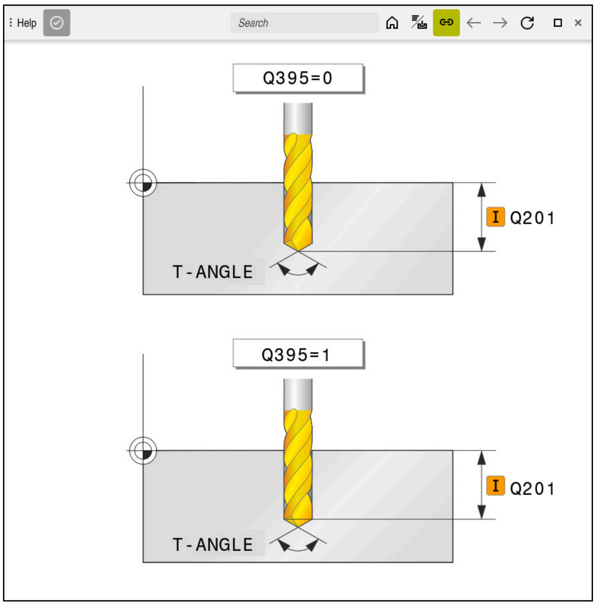
- **Help** application
  Further information: "Help application", Page 83
- User's Manual as the **TNCguide** integrated product aid
  Further information: "User's Manual as integrated product aid: TNCguide", Page 82

Description of function

The **Help** workspace can be selected in the **Editor** operating mode and in the **MDI** application.

Further information: "Editor operating mode", Page 216

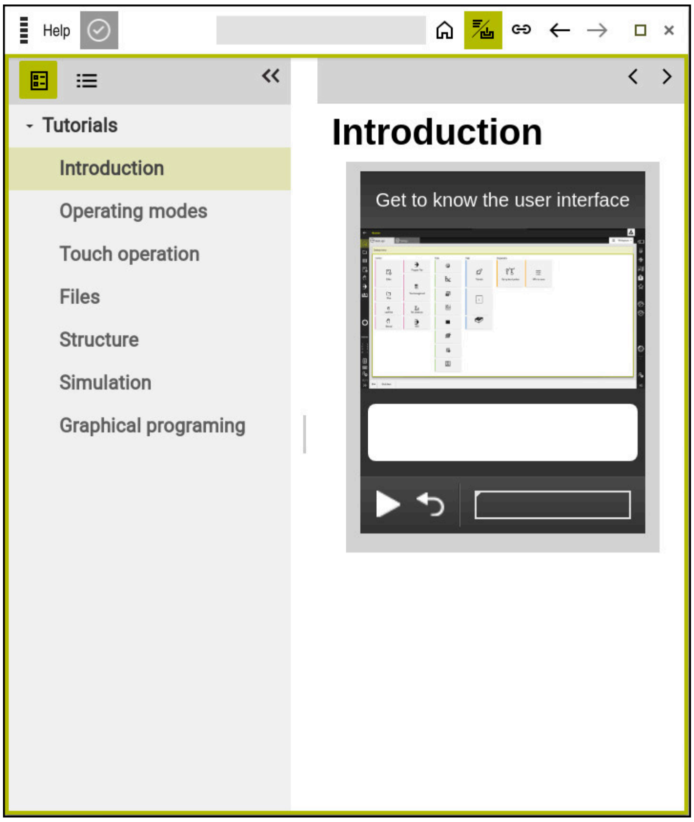
Further information: "The MDI Application ", Page 1933



Help workspace with a help graphic for a cycle parameter

If the **Help** workspace is active, the control can display the help screen in it during programming instead of in the **Program** workspace.

Further information: "Program workspace", Page 217



Help workspace with opened **TNCguide**

When the **Help** workspace is active, the control can display the integrated **TNCguide** product aid.

Further information: "User's Manual as integrated product aid: TNCguide", Page 82

Symbols in the Help workspace

Symbol	Function
	Show start page The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., **TNCguide**). If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function. **Further information:** "Symbols", Page 84
	Displaying the **TNCguide** **Further information:** "User's Manual as integrated product aid: TNCguide", Page 82
	Displaying help images during programming

28.1.1 Note

Use the machine parameter **stdTNChelp** (no. 105405) to define whether the control displays help graphics as pop-up windows in the **Program** workspace.

Further information: "Program workspace", Page 217

28.2 Virtual keyboard of the control bar

Application

You can use the virtual keyboard for entering NC functions, letters, and numbers, and for navigation.

The virtual keyboard offers the following modes:

- NC input
- Text input
- Formula entry

Description of function

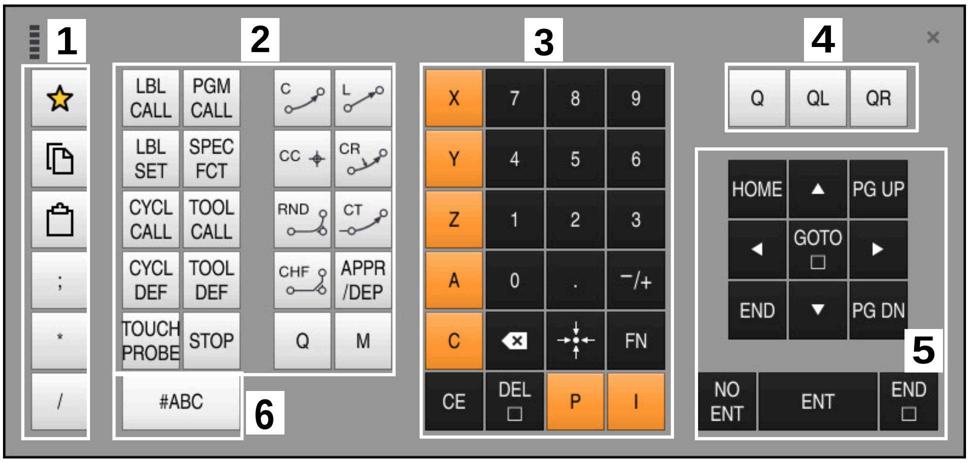
The control opens NC input mode by default after the start procedure.

You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

The control remembers the position and mode of the virtual keyboard until it is shut down.

The **Keyboard** workspace provides the same functions as the virtual keyboard.

NC input areas



Virtual keyboard in NC input mode

NC input mode contains the following areas:

1 File functions
- Define favorites
- Copy
- Paste
- Add comment
- Add structure item
- Hide NC block

2 NC functions

3 Axis keys and numerical input

4 Q parameters

5 Navigation and dialog keys

6 Switch to text input

If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:
- **Q**
- **QL**
- **QR**

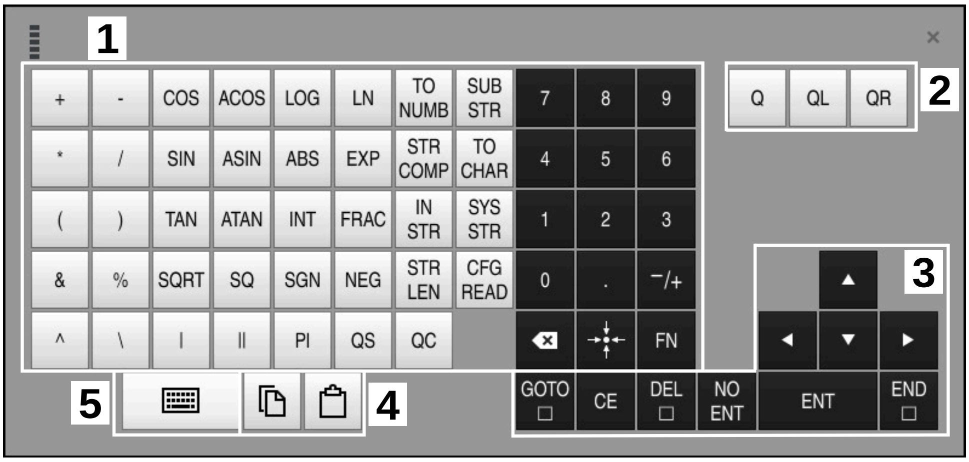
Areas of text input

Virtual keyboard in text input mode

The text input contains the following areas:

1 Input
2 Navigation and dialog keys
3 Copying and pasting
4 Switch to formula input

Areas of formula input



Virtual keyboard in formula input mode

The formula input contains the following areas:

1 Input
2 Q parameters
3 Navigation and dialog keys
4 Copying and pasting
5 Switch to NC input

28.2.1 Opening and closing the virtual keyboard

To open the virtual keyboard:

- Select the **virtual keyboard** on the control bar
- > The control opens the virtual keyboard.

To close the virtual keyboard:

- Select the **virtual keyboard** when the virtual keyboard is open
- Or press **Close** in the virtual keyboard
- > The control closes the virtual keyboard.

28.3 GOTO function

Application

With the **GOTO** key or the **GOTO block number** button you define an NC block at which the control positions the cursor. In the **Tables** mode you use the **GOTO record** button to define a table row.

Description of function

If an NC program is open for simulation or execution, the control additionally positions the execution cursor in front of the NC block. The control then starts program run or the simulation beginning from the defined NC block without considering the preceding lines of the NC program.

You can enter the block number directly or find it in the NC program with the **Search** function.

28.3.1 Selecting an NC block with GOTO

To select an NC block:

GOTO

- Select **GOTO**
- > The control opens the **GOTO jump instruction** window.
- Enter the block number

OK

- Press **OK**
- > The control positions the cursor to the defined NC block.

NOTICE

Danger of collision!

If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions, e.g. transformations. This means that there is a risk of collision during subsequent traversing movements!

- Use **GOTO** only when programming and testing NC programs
- Only use **Block scan** when executing NC programs

Further information: "Block scan for mid-program startup", Page 1965

Notes

- You can also use the keyboard shortcut **CTRL+G** instead of the **GOTO** button.
- If the control in the action bar shows an icon for selection, you can open the selection window with **GOTO**.

28.4 Adding comments

Application

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

Description of function

You have the following possibilities for adding comments:

- Comment within an NC block
- Comment as a separate NC block
- Define existing NC block as comment

The control marks comments with a preceding **;** character. The control does not execute comments during simulation or program run.

A comment may contain up to 255 characters.

> The last character in a comment block must not be a tilde sign (**~**).

28.4.1 Adding a comment as an NC block

To add a comment as a separate NC block:

- Select the NC block after which the comment is to be added

;

- Select **;**
- \> After the selected NC block, the control adds a comment as a new NC block.
- Define the comment

28.4.2 Adding a comment in an NC block

To add a comment within an NC block:

- Edit the desired NC block

;

- Select **;**
- \> The control inserts a **;** character at the end of the block.
- Define the comment

28.4.3 Commenting an NC block out or in

Use the **Comment out/in** button to define an existing NC block as a comment or to change a comment back to an NC block.

To comment an existing NC block in or out:

- Select the desired NC block

- Select **Comment Off/On**
- > The control inserts a **;** character at the beginning of the block.
- > If the NC block is already defined as a comment, the control removes the **;** character.

28.5 Hiding NC blocks

Application

Use **/** or the **Skip block off/on** button to hide NC blocks.

If you hide NC blocks, you can skip the hidden NC blocks in program run.

Related topics

- **Program Run** operating mode

  Further information: "Program Run operating mode", Page 1954

Description of function

If you mark an NC block with a **/** character, then the NC block is hidden. If you activate the **Skip block** toggle switch in the **Program Run** operating mode or in the **MDI** application, the control skips the NC block during execution.

If the toggle switch is active, the control grays out the NC blocks to be skipped.

Further information: "Icons and buttons", Page 1956

28.5.1 Hiding or showing NC blocks

To hide or show an NC block:

- Select the desired NC block

- Select **Skip block off/on**
- > The control adds a **/** character before the NC block.
- > If the NC block is already hidden, the control removes the **/** character.

28.6 Structuring of NC programs

Application

You can use structure items to make long and complex NC programs more clear and legible, and also to navigate more quickly through an NC program.

Related topics

- **Structure** column of the **Program** workspace
  Further information: "Structure column in the Program workspace", Page 1514

Description of function

You can use structure items to arrange your NC programs. Structure items are texts that you can use as comments or headlines for the subsequent program lines.

A structure item may contain up to 255 characters.

The control displays the structure items in the **Structure** column.

Further information: "Structure column in the Program workspace", Page 1514

28.6.1 Adding a structure item

To insert a structure item:

- Select the NC block after which you want to add the structure item

*

- Select *
- After the selected NC block, the control adds a structure item as a new NC block.
- Define the structure text

28.7 Structure column in the Program workspace

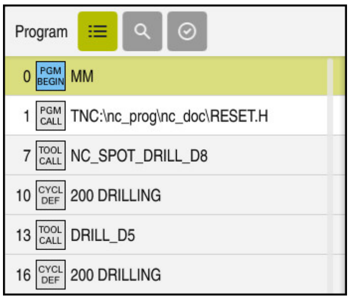
Application

When you open an NC program, the control searches the NC program for structure elements and displays these structure elements in the **Structure** column. The structure elements act like links and thus enable fast navigation in the NC program.

Related topics

- **Program** workspace, defining contents of the **Structure** column
  Further information: "Settings in the Program workspace", Page 220
- Inserting structure items manually
  Further information: "Structuring of NC programs", Page 1514

Description of function



Structure column with automatically created structure elements

When you open an NC program, the control automatically creates the structure.

In the **Program settings** window, you define which structure elements the control displays in the structure. The **PGM BEGIN** and **PGM END** structure elements cannot be hidden.

Further information: "Settings in the Program workspace", Page 220

The **Structure** column shows the following information:

- NC block number
- Icon of the NC function
- Function-dependent information

The control displays the following icons within the structure:

Icon	Syntax	Information
PGM BEGIN	**BEGIN PGM**	Unit of measurement of the NC program **MM** or **INCH**
TOOL CALL	**TOOL CALL**	■ Name or number of the tool, if applicable ■ Index of the tool, if applicable ■ Comment, if applicable
*	*** Structure block**	■ Entered string, if applicable ■ Comment, if applicable
LBL SET	**LBL SET**	■ Name or number of the label ■ Comment, if applicable
LBL SET	**LBL 0**	■ Number of the label ■ Comment, if applicable
CYCL DEF	**CYCL DEF**	Number and name of the defined cycle
TCH PROBE	**TCH PROBE**	Number and name of the defined cycle
MON START	**MONITORING SECTION START**	■ String entered in the **AS** syntax element, if applicable ■ Comment, if applicable
MON STOP	**MONITORING SECTION STOP**	Comment, if applicable
PGM CALL	**PGM CALL**	■ Path of the called NC program (e.g., **TNC:\Safe.h**) ■ Comment, if applicable

Icon	Syntax	Information
SPEC FCT	**FUNCTION MODE**	■ Selected machining mode: **MILL**, **TURN**, or **GRIND** ■ Selected kinematics, if applicable ■ Comment, if applicable
M2 M30	**M2** or **M30**	Comment, if applicable
M1	**M1**	Comment, if applicable
STOP M0	**STOP** or **M0**	Comment, if applicable
APPR	**APPR**	■ Selected approach function ■ Comment, if applicable
DEP	**DEP**	■ Selected departure function ■ Comment, if applicable
PGM END	**PGM END**	No additional information

In the **Program Run** operating mode the **Structure** column contains all structure items, including those of the called NC programs. The control indents the structure of the called NC programs.

Further information: "Navigation path in the Program workspace", Page 1962

> The control displays comments as separate NC blocks, rather than including them in the structure. These NC blocks start with a semicolon character (**;**).
>
> "Adding comments"

28.7.1 Editing an NC block using the structure

To edit an NC block using the structure:

- Open an NC program
- Open the **Structure** column
- Select structure element
- > The control positions the cursor on the corresponding NC block in the NC program. The focus of the cursor remains in the **Structure** column.
- Select the right arrow
- > The focus of the cursor changes to the NC block.
- Select the right arrow
- > The control edits the NC block.

Notes

- In the case of long NC programs, establishing the structure may take longer than loading the NC program. Even if the structure has not yet been created, you can still work independently of it in the loaded NC program.
- You can navigate within the **Structure** column using the up and down arrow keys.
- If you mark structuring items in the **Structure** column, the control propagates the marking to the corresponding NC blocks in the NC program. Use the **CTRL+SPACE** key shortcut to stop marking. If you press **CTRL+SPACE** again, the control restores the marked selection.
- The control shows called NC programs in the structure with a white background. If you double-tap or click on such a structure element, the control opens the NC program if necessary in a new tab. When the NC program is open, the control switches to the corresponding tab.

28.8 Search column in the Program workspace

Application

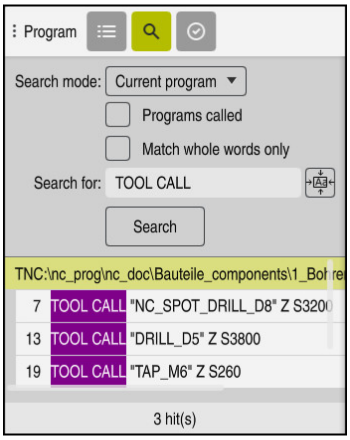
In the **Search** column, you can search the NC program for any character strings, such as individual syntax elements. The control lists all the results found.

Related topics

- Search for the same syntax element in the NC program with the arrow keys

  Further information: "Searching for the same syntax elements in different NC blocks", Page 226

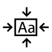
Description of function



Search column in the **Program** workspace

The control provides the full range of functions in the **Editor** operating mode only. In the **MDI** application, you can search only the active NC program. The **Search and replace** mode is not available in the **Program Run** operating mode.

The control provides the following functions, icons and buttons in the **Search** column:

Area	Function
Search in:	▪ **Current program** Search the current NC program and optionally all called NC programs ▪ **Opened programs** Browse all open NC programs ▪ **Search and replace** Search for strings and replace them with new strings, such as syntax elements **Further information:** "Search and replace mode", Page 1519
Match whole words only	If you select the check box, the control only displays exact matches. This means that if you search for **Z+10**, for example, the control ignores **Z+100**. The check box is available in every mode.
Search for:	In the input area, you define the search term. If you have not yet entered any characters, the control suggests the last six search terms for selection. The search is not case-sensitive.
	Use the **Apply selection** icon to transfer the currently selected syntax element to the input area. If the selected NC block is not edited, the control accepts the syntax initiator.
Search	Use this button to start the search in the **Current program** and **Opened programs** modes.

The control shows the following information about the results:

- Number of results
- File paths of the NC programs
- NC block numbers
- Entire NC blocks

The control groups the results according to NC programs. If you select a result, the control positions the cursor on the corresponding NC block.

Search and replace mode

In **Search and replace** mode, you can search for strings and replace the results found with other strings, such as syntax elements.

The control performs a syntax check before replacing a syntax element. With the syntax check, the control ensures that the new content results in correct syntax. If the result produces a syntax error, the control does not replace the content and displays a message.

In **Search and replace** mode, the control provides the following checkboxes and buttons:

Checkbox or button	Meaning
Search backward	The control searches the NC program from bottom to top.
Wrap around	The control searches the entire NC program, beyond the start and end of the NC program.
Find next	The control searches the NC program for the search term. The control marks the next result in the NC program.
Replace	The control performs a syntax check and replaces the selected content in the NC program with the content of the **Replace with:** field.
Replace and find next	If a search has not yet been performed, the control only marks the first result. When a result is highlighted, the control performs a syntax check and automatically replaces the found content with the contents of the **Replace with:** field. The control then marks the next result.
Replace all	The control performs a syntax check and automatically replaces all found results with the contents of the **Replace with:** field.

28.8.1 Search for and replace syntax elements

To search for and replace syntax elements in the NC program:

- Select an operating mode, e.g. **Editor**
- Select the desired NC program
- > The control opens the selected NC program in the **Program** workspace.

- Open the **Search** column
- In the **Search in:** field, select the **Search and replace** function
- > The control displays the **Search for:** and **Replace with:** fields.
- In the **Search for:** field, enter the search content, e.g. **M4**
- In the **Replace with:** field, enter the desired content, e.g. **M3**

- Select **Find next**
- > The control highlights the first result in the NC program in purple.

- Select **Replace**
- > The control performs a syntax check and replaces the content if the check is successful.

Notes

- The search results are retained until you shut down the control or search again.
- If you double-tap or click on a search result in a called NC program, the control opens the NC program (on a new tab if not already open). If the NC program is already open, the control switches to the corresponding tab.
- If you have not entered a value for **Replace with:**, the control deletes the search value.

28.9 Program comparison

Application

Use the **Program comparison** function to determine differences between two NC programs. You can transfer the deviations to the active NC program. If there are unsaved changes in the active NC program, you can compare the NC program with the last saved version.

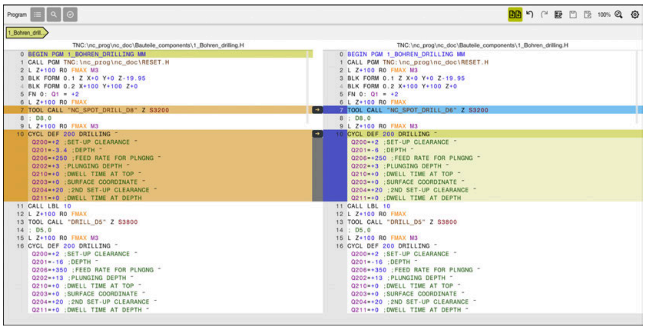
Requirements

- Max. 30,000 lines per NC program

  The control takes into account the actual lines, not the number of NC blocks. Some NC blocks, particularly those consisting of cycles, can contain several lines within one block number.

  Further information: "Contents of an NC program", Page 212

Description of function



Program comparison of two NC programs

You can use the program comparison only in the **Editor** operating mode in the **Program** workspace.
The control shows the active NC program on the right and the comparison program on the left.
The control marks differences with the following colors:

Color	Syntax element
Gray	Missing NC block or missing line for NC functions of different length
Orange	NC block with difference in comparison program
Blue	NC block with difference in the active NC program

During the program comparison, you can edit the active NC program, but not the comparison program.
If NC blocks differ, you can use an arrow symbol to transfer the NC blocks of the comparison program to the active NC program.

28.9.1 Applying differences to the active NC program

To transfer differences to the active NC program:

- Select the **Editor** operating mode
- Open an NC program
- Select **Program comparison**
- > The control opens a pop-up window for file selection.
- Select comparison program

Select

- Select **Select**
- > The control shows both NC programs in the comparison view and marks all differing NC blocks.
- Select the arrow symbol for the desired NC block
- > The control transfers the NC block to the active NC program.
- Select **Program comparison**
- > The control closes the comparison view and transfers the differences to the active NC program.

Notes

- If the compared NC programs contain more than 1000 differences, the control cancels the comparison.
- If an NC program contains unsaved changes, the control displays an asterisk in front of the name of the NC program in the tab of the application bar.
- If you mark multiple NC blocks in the comparison program, you can apply those NC blocks simultaneously. If you mark multiple NC blocks in the active NC program, you can overwrite those NC blocks simultaneously.

  Further information: "Context menu", Page 1522

28.10 Context menu

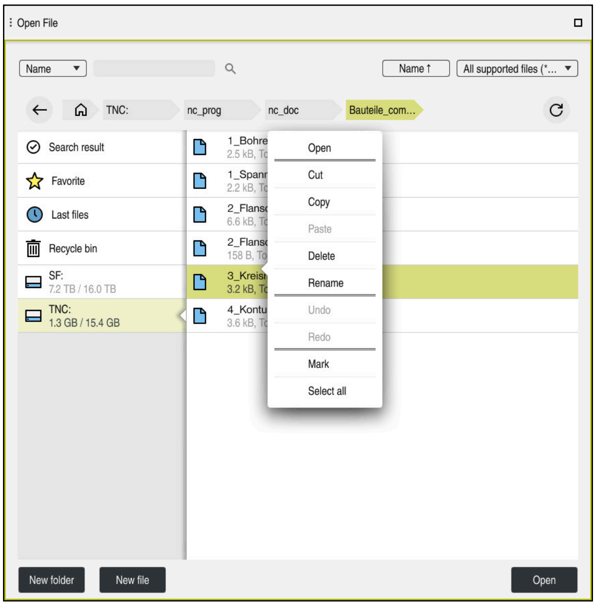
Application

With a long-press gesture or by right-clicking with the mouse, the control opens a context menu for the selected element, such as an NC block or file. Use the various functions of the context menu to run commands that affect the currently selected element(s).

Description of function

The functions available in the context menu depend on the selected element as well as the selected operating mode.

General



Context menu in the **Open File** workspace

The context menu offers the following functions:

- **Cut**
- **Copy**
- **Paste**
- **Delete**
- **Undo**
- **Redo**
- **Mark**
- **Select all**

> If you select the **Mark** or **Select all** functions, the control opens the action bar. The action bar displays all functions that are currently available for selection from the context menu.

As an alternative to the context menu, you can use keyboard shortcuts:

Further information: "Icons on the control's user interface", Page 123

Key or keyboard shortcut	Meaning
CTRL+BLANK	Mark the selected line
SHIFT+↑	Additionally mark a line above it
SHIFT+↓	Additionally mark a line below it
SHIFT+ PG UP	Mark from the cursor position to the beginning of the page Not in the **Tables** operating mode
SHIFT+ PG DN	Mark from the cursor position to the end of the page Not in the **Tables** operating mode
SHIFT+ HOME	Mark from the cursor position to the first row Not in the **Tables** operating mode
SHIFT+ END	Mark from the cursor position to the last row Not in the **Tables** operating mode
ESC	Cancel marking

> These keyboard shortcuts do not work in the **Job list** workspace.

Context menu in the Files operating mode

In the **Files** operating mode, the context menu also offers the following functions:

- **Open**
- **Select in Program Run**
- **Rename**

For the navigation functions, the context menu offers the respectively relevant functions, such as **Discard search results**.

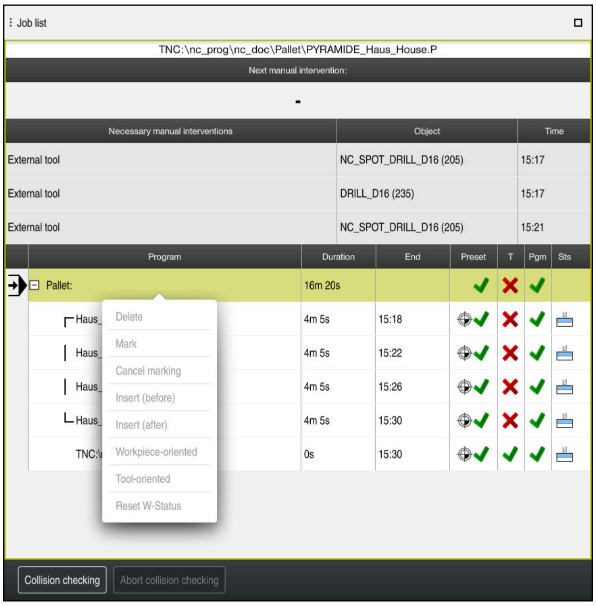
Further information: "Context menu", Page 1522

Context menu in the Tables operating mode

In the **Tables** operating mode the context menu additionally offers the **Cancel** function. Use the **Cancel** function to abort the marking action.

Further information: "Tables operating mode", Page 1980

Context menu in the Job list (option 22) workspace



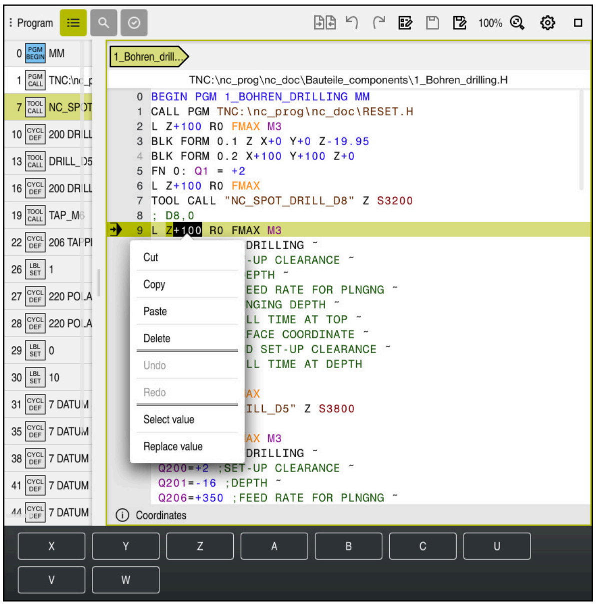
Context menu in the **Job list** workspace

In the **Job list** workspace, the context menu offers the following additional functions:

- **Cancel marking**
- **Insert (before)**
- **Insert (after)**
- **Workpiece-oriented**
- **Tool-oriented**
- **Reset W-Status**

Further information: "Job list workspace", Page 1938

Context menu in the Program workspace



Context menu for selected value in the **Program** workspace of the **Editor** operating mode

In the **Program** workspace, the context menu offers the following additional functions:

- **Insert last NC block**

  This function allows you to insert the most recently deleted or edited NC block. You can insert this NC block in any desired NC program.

  Only in the **Editor** operating mode and in the **MDI** application
- **Create NC sequence**

  Only in the **Editor** operating mode and in the **MDI** application

  Further information: "NC sequences for reuse", Page 392
- **Edit contour**

  Only in the **Editor** operating mode

  Further information: "Importing contours into graphical programming", Page 1446
- **Select value**

  Active when you select a value of an NC block.
- **Replace value**

  Active when you select a value of an NC block.

Further information: "Program workspace", Page 217

> The **Select value** and **Replace value** functions are only available in the **Editor** operating mode and in the **MDI** application.
>
> **Replace value** is also available during editing. In this case the otherwise necessary marking of the value to be replaced is omitted.
>
> For example, you can copy values from the calculator or position display to the clipboard and then paste them with the **Replace value** function.

Further information: "Calculator", Page 1527

Further information: "Status overview on the TNC bar", Page 169

If you select an NC block, the control displays marker arrows at the beginning and end of the selected area. Use these marker arrows to change the highlighted area.

Context menu in the configuration editor

In the configuration editor, the context menu also provides the following functions:

- **Direct entry of values**
- **Create copy**
- **Restore copy**
- **Change key name**
- **Open element**
- **Remove element**

Further information: "Machine parameters", Page 2152

28.11 Calculator

Application

The control offers a calculator on the control bar. You can copy the result to the clipboard and also paste values from the clipboard.

Description of function

The calculator offers the following functions:

- Basic mathematical operations
- Basic trigonometric functions
- Square root
- Exponential calculation
- Reciprocal value

Calculator

You can switch between the radian **RAD** or degrees **DEG** modes.

You can copy the result to the clipboard as well as paste the last stored value from the clipboard to the calculator.

The calculator saves the last ten calculations in the history. You can use these saved results for further calculations. You can clear the history manually.

28.11.1 Opening and closing the calculator

To open the calculator:

- ▶ Select the **calculator** on the control bar
- > The control opens the calculator.

To close the calculator:

- ▶ Select the **calculator** when the calculator is open
- > The control closes the calculator.

28.11.2 Selecting a result from the history

To select a result from the history for further calculations:

- Select **History**
- > The control opens the calculator's history.
- Select the desired result
- Select **History**
- > The control closes the calculator's history.

28.11.3 Deleting the history

To delete the calculator's history:

- Select **History**
- > The control opens the calculator's history.
- Select **Delete**
- > The control deletes the calculator's history.

28.12 Cutting data calculator

Application

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

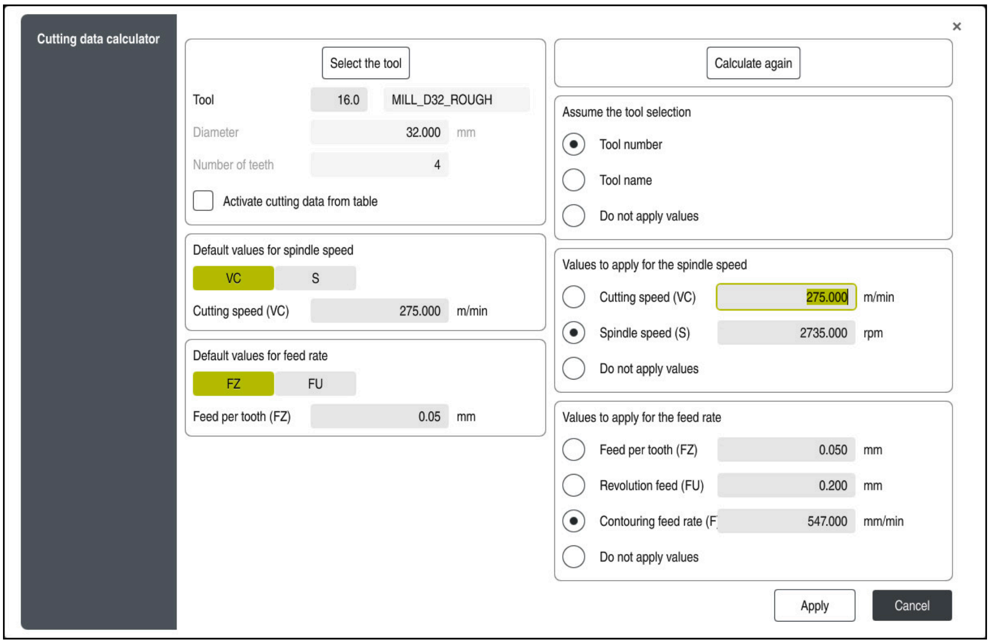
The control offers the **OCM cutting data calculator** for OCM cycles (option 167).

Further information: "OCM Cutting data calculator (option 167)", Page 669

Requirement

- Milling operation **FUNCTION MODE MILL**

Description of function



Window for the **Cutting data calculator**

On the left side of the cutting data calculator you enter the information. On the right side the control displays the calculated results.

If you select a tool defined in the tool management, the control automatically applies the tool diameter and number of teeth.

You can calculate the spindle speed as follows:

- Cutting speed **VC** in m/min
- Spindle speed **S** in rpm

You can calculate the feed rate as follows:

- Feed per tooth **FZ** in mm
- Feed per revolution **FU** in mm

Or you can use tables to calculate the cutting data.

Further information: "Calculation with tables", Page 1530

Applying values

After the cutting data have been calculated, you can specify which values the control should apply.

You can choose among the following options for the tool:

- **Tool number**
- **Tool name**
- **Do not apply values**

You can choose among the following for the spindle speed:

- **Cutting speed (VC)**
- **Spindle speed (S)**
- **Do not apply values**

You can choose among the following for the feed rate:

- **Tooth feed (FZ)**
- **Revolution feed (FU)**
- **Contouring feed rate (F)**
- **Do not apply values**

Calculation with tables

You must define the following in order to calculate the cutting data with tables:

- Workpiece material in the table **WMAT.tab**

  Further information: "Table for workpiece materials WMAT.tab", Page 2048
- Tool cutting material in table **TMAT.tab**

  Further information: "Table for tool materials TMAT.tab", Page 2048
- Combination of workpiece material and cutting material in the cutting data table ***.cut** or in the diameter-dependent cutting data table ***.cutd**

> 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**).
>
> **Further information:** "Cutting data table *.cut", Page 2049
>
> 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 *.cutd", Page 2050

- Parameters of the tool in tool management:
  - **R**: Tool radius
  - **LCUTS**: Number of cutting edges
  - **TMAT**: Cutting material from **TMAT.tab**
  - **CUTDATA**: Table row from the ***.cut** or ***.cutd** cutting data table

28.12.1 Opening the cutting data calculator

To open the cutting data calculator:

- ▶ Edit the desired NC block
- ▶ Select the syntax element for the feed rate or spindle speed
- ▶ Select **Cutting data calculator**
- > The control opens the **Cutting data calculator** window.

28.12.2 Calculating the cutting data with tables

The following prerequisites must be fulfilled in order to calculate the cutting data with tables:

- The **WMAT.tab** table exists
- The **TMAT.tab** table exists
- The ***.cut** or ***.cutd** table exists
- Tool material and cutting data table are assigned in the tool management

To calculate the cutting data with tables:

- Edit the desired NC block
- Open the **Cutting data calculator**
- Select **Activate cutting data from table**
- Use **Select material** to choose the workpiece material
- Use **Select type of machining** to choose the combination of workpiece material and tool material
- Select the desired values to be applied

Apply

- Press **Apply**
- > The control applies the calculated values in the NC block.

Notes

You cannot calculate the cutting data in turning mode (option 50) because the feed rate and spindle speed data are different in turning mode from milling mode.

In turning operations the feed rates are often defined in millimeters per revolution (mm/1) (**M136**), whereas the cutting data calculator always calculates feed rates in millimeters per minute (mm/min). Furthermore, the radius in the cutting data calculator is referenced to the tool; turning operations, however, require the workpiece diameter.

28.13 Message menu on the information bar

Application

In the message menu of the information bar, the control shows pending errors and notes. When opened, the control displays detailed information about the messages.

Description of function

The control uses the following symbols to differentiate between the types of messages:

Symbol	Message type	Meaning
	Error Question type	The control displays a dialog with several options you can select from. You cannot clear this error message: you can only choose one of the possible responses. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.
	Reset error	The control must be restarted. This message cannot be cleared.
	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	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.
	Information	You can continue without clearing the message. You can clear the information at any time.
	Note	You can continue without clearing the message. The control displays the note until you press the next valid key.
		No pending messages

The message menu is collapsed by default.

The control displays messages upon various events, for example:

- Logical errors in the NC program
- Impossible contour elements
- Improper touch-probe inserts
- Hardware updates

Content

Collapsed message menu on the information bar

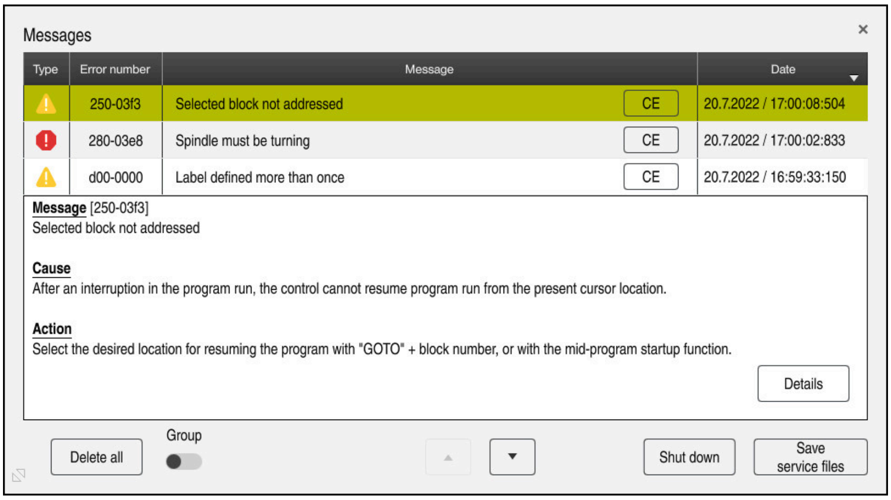
When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

The control displays the following information in the collapsed message menu:

- Message type
- Message
- Quantity of pending errors, warnings, and informational messages

Detailed messages

If you tap or click the symbol or within the message, the control expands the message menu.



Expanded message menu with pending messages

The control displays all pending messages in chronological order.

The message menu shows the following information:

- Message type
- Error number
- Message
- Date
- Additional information (root cause, correction, information on the NC program)

Deleting messages

Messages can be deleted in the following ways:

- **CE** key
- **CE** button in the message menu
- **Delete all** button in the message menu

Details

Press the **Details** button to show or hide internal information about the message. This information is of importance in case servicing is necessary.

Group

If you activate the **Group** toggle switch, the control displays all messages with the same error number in one row. This makes the list of messages shorter and easier to read.

Under the error number, the control displays the quantity of messages. Use **CE** to clear all messages of a group.

Service file

Click the **Save service files** button to open the **Save service files** window.

In the **Save service files** window, you can create service files in the following ways:

- If an error occurs, you can create a service file manually.

  Further information: "Creating a service file manually", Page 1534
- If an error occurs repeatedly, a service file can be created automatically by means of the error number. Once the respective error occurs, the control saves a service file.

  Further information: "Creating a service file automatically", Page 1534

Service files help service technicians in troubleshooting the problem. The control saves data that provide information about the current machine and operation status, such as active NC programs up to 10 MB, tool data, and keystroke logs.

28.13.1 Creating a service file manually

To create a service file manually:

- ▶ Expand the message menu

Save service files

- ▶ Select **Save service files**
- > The control opens the **Save service file** window.
- ▶ Enter the file name

OK

- ▶ Press **OK**
- > The control saves the service file in the **TNC:\service** directory.

28.13.2 Creating a service file automatically

You can specify up to five error numbers for which the control will automatically create a service file if one of these errors occurs.

To specify a new error number:

- ▶ Expand the message menu

Save service files

- ▶ Select **Save service files**
- > The control opens the **Save service file** window.

Setting for autosave

- ▶ Select **Setting for autosave**
- > The control opens a table of error numbers.
- ▶ Enter the desired error number
- ▶ Activate the **Active** check box
- > If the error occurs, the control automatically creates a service file.
- ▶ Enter a comment, if applicable (e.g., to describe the problem)

29

Simulation Workspace

29.1 Fundamentals

Application

In the **Editor** operating mode, you can use the **Simulation** workspace to graphically test whether NC programs are programmed correctly and run without collisions.

In the **Manual** and **Program Run** operating modes, the control shows the current traverse motions of the machine in the **Simulation** workspace.

Requirements

- Tool definitions according to the tool data from the machine
- Workpiece blank definition that is valid for a test run

  Further information: "Defining a workpiece blank with BLK FORM", Page 258

Description of function

In the **Editor** operating mode, the **Simulation** workspace can be open for only one NC program at a time. If you want to open the workspace on a different tab, the control prompts you for confirmation.

The functions available in the simulation depend on the following settings:

- Selected type of model, for example **2.5D**
- Selected quality of model, for example **Medium**
- Selected mode, for example **Machine**

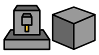
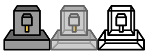
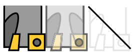
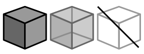
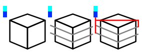
Icons in the Simulation workspace

The following symbols are shown in the **Simulation** workspace:

Symbol	Function
	Visualization options **Further information:** "Visualization options column", Page 1537
	Workpiece options **Further information:** "Workpiece options column", Page 1539
	Pre-defined views **Further information:** "Pre-defined views", Page 1545
	Export simulated workpiece as STL file **Further information:** "Exporting a simulated workpiece as STL file", Page 1546
	Simulation settings **Further information:** "Simulation settings window", Page 1541
	Status of dynamic collision monitoring (DCM) in the simulation **Further information:** "Visualization options column", Page 1537
✓++	Status of the **Advanced checks** function **Further information:** "Visualization options column", Page 1537
	Selected quality of model **Further information:** "Simulation settings window", Page 1541
T 0	Number of the active tool
00:00:00	Current program run-time

Visualization options column

In the **Visualization options** column you can define the following display modes and functions:

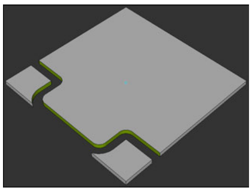
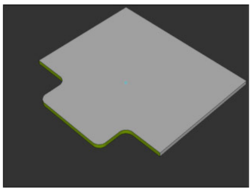
Symbol or switch	Function	Requirements
	Select the **Machine** or **Workpiece** mode If you select the **Machine** mode, the control displays the defined workpiece, the collision objects, and the tool. In the **Workpiece** mode the control shows the workpiece to be simulated. Depending on the selected mode, different functions are available.	
Workpiece position	Use this function to define the position of the workpiece preset for the simulation. You can use a button to select a workpiece preset from the preset table. **Further information:** "Preset management", Page 1025	■ **Machine** mode ■ Type of model: **2.5D**
	You can select between the following display modes for the machine: ■ **Original**: Shaded, opaque representation ■ **Semitransparent**: Transparent representation ■ **Wire-frame model**: Representation of the machine contours	■ **Workpiece** mode ■ Type of model: **2.5D**
	You can select between the following display modes for the tool: ■ **Original**: Shaded, opaque representation ■ **Semitransparent**: Transparent representation ■ **Invisible**: The object is hidden	■ **Workpiece** mode ■ Type of model: **2.5D**
	You can select between the following display modes for the workpiece: ■ **Original**: Shaded, opaque representation ■ **Semitransparent**: Transparent representation ■ **Invisible**: The object is hidden	■ **Workpiece** mode ■ Type of model: **2.5D**
	You can show the tool paths during the simulation. The control displays the center-line path of the tools. You can choose between the following display modes for the tool paths: ■ **None**: Do not show tool paths ■ **Feed**: Show tool paths with programmed feed rate ■ **Feedrate + FMAX**: Show tool paths with programmed feed rate and with programmed rapid traverse	■ **Workpiece** mode ■ Operating mode: **Editor**
Clamping situation	Use this toggle switch to show the worktable and fixture, if required.	■ **Workpiece** mode ■ Type of model: **2.5D**
DCM	Use this toggle switch to activate or deactivate collision monitoring (DCM, option 40) for the simulation. **Further information:** "Dynamic Collision Monitoring (DCM) in the Editor operating mode", Page 1167	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**

Symbol or switch	Function	Requirements
Advanced checks	Use this toggle switch to activate the **Advanced checks** function. **Further information:** "Advanced checks in the simulation", Page 1190	■ Operating mode: **Editor**
Breakpoints	If you activate this toggle switch, the control opens the **Breakpoints** window with the following selection possibilities: ■ **Skip block** If an NC block is preceded by a **/** character, then the NC block is hidden. If you activate the **Skip block** toggle switch, the control skips all hidden NC blocks in the simulation. **Further information:** "Hiding NC blocks", Page 1513 If the toggle switch is active, the control grays out the NC blocks to be skipped. **Further information:** "Appearance of the NC program", Page 219 ■ **Pause at M1** If you activate the toggle switch, the control pauses the simulation at each **M1** miscellaneous function in the NC program. **Further information:** "Overview of miscellaneous functions", Page 1319 If this toggle switch is inactive, the control grays out the **M1** syntax element. **Further information:** "Appearance of the NC program", Page 219	■ Operating mode: **Editor**

Workpiece options column

In the **Workpiece options** column you can define the following simulation functions for the workpiece:

Switch or button	Function	Requirements
Measuring	Use this function to measure any points on the simulated workpiece. **Further information:** "Measuring function", Page 1548	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Cutout view	Use this function to cut through the simulated workpiece along a plane. **Further information:** "Cutout view in the simulation", Page 1549	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Highlight workpiece edges	Use this function to highlight the edges of the simulated workpiece.	■ **Workpiece** mode ■ Type of model: **2.5D**
Workpiece blank frame	The control uses this function to show the outside lines of the workpiece blank.	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Finished part	Use this function to show a finished part that was defined with the help of the **BLK FORM FILE** function. **Further information:** "Cutout view in the simulation", Page 1549	■ Type of model: **2.5D**
Software limit switches	Use this function to activate the software limit switches of the machine for the active traverse range in the simulation. By simulating the limit switches you can check whether the working space of the machine is sufficient for the simulated workpiece. **Further information:** "Simulation settings window", Page 1541	■ Operating mode: **Editor**

Switch or button	Function	Requirements
Workpiece coloring	■ **Grayscale** The control displays the workpiece in various shades of gray. ■ **Tool based** The control displays the workpiece in color. Each cutting tool is assigned a separate color. ■ **Model comparison** The control displays a comparison between the workpiece blank and the finished part. **Further information:** "Model comparison", Page 1551 ■ **Monitoring** The control displays a heat map on the workpiece: ■ Component heat map with **MONITORING HEAT MAP** **Further information:** "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1230 **Further information:** "Cycles for monitoring", Page 1231 ■ Process heat map with **SECTION MONITORING** **Further information:** "Process Monitoring (option 168)", Page 1236	■ Type of model: **2.5D** ■ **Model comparison** function only in **Workpiece** mode ■ **Monitoring** function only in the **Program Run** operating mode
Reset the workpiece	Use this function to reset the workpiece back to the workpiece blank	■ Operating mode: **Editor** ■ Type of model: **2.5D**
Reset the tool paths	Use this function to reset the simulated tool paths.	■ **Workpiece** mode ■ Operating mode: **Editor**
Remove the chips	Use this function to remove from the simulation those parts of the workpiece that were cut off during machining.  Workpiece before clean-up  Workpiece after clean-up	■ Operating mode: **Editor** ■ Type of model: **3D**

Simulation settings window

The **Simulation settings** window is available only in the **Editor** operating mode.

The **Simulation settings** window consists of the following areas:

<table>
<tr><th>Area</th><th>Function</th></tr>
<tr><td>General</td><td>

Model type

None: fast line graphics without volume model
2.5D: quick 3D representation without undercuts
3D: realistic 3D representation with undercuts

Quality

Low: low-quality model, low memory use
Medium: normal-quality model, average memory use
High: high-quality model, uses much memory
Highest: best-quality model, uses very much memory

Mode

Milling
Turning
Grinding

Active kinemat.
Select the kinematics model for the simulation from a selection menu. The machine manufacturer enables the kinematics models.
Generate tool-usage file

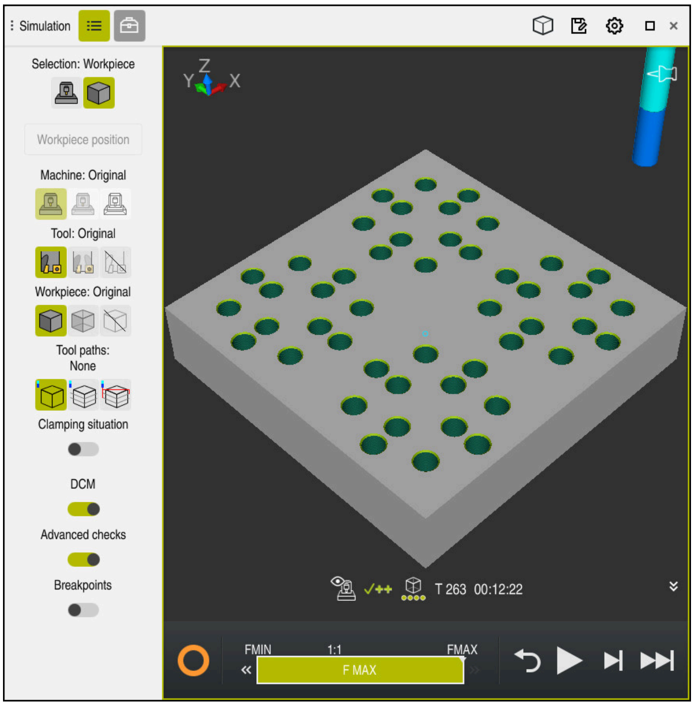
Never
Do not generate a tool-usage file
Once
Generate a tool-usage file for the next simulated NC program
Always
Generate a tool-usage file for every simulated NC program

Further information: "Channel settings", Page 2104</td></tr>
<tr><td>Traverse ranges</td><td>

Traverse ranges
In this selection menu you can choose one of the traverse ranges defined by the machine manufacturer, such as Limit1. In each traverse range the machine manufacturer defines different software limit switches for each axis of the machine. For example, the machine manufacturer defines traverse ranges for large machines with two separate working spaces.
Further information: "Workpiece options column", Page 1539
Active traverse ranges
This function shows the active traverse range and the values defined for within that range.
</td></tr>
</table>

Area	Function
Tables	You can select tables specifically for the **Editor** operating mode. The control uses the selected tables for the simulation. The selected tables are independent of any tables that are active in other operating modes. You use a selection menu to choose the tables. You can select the following tables for the **Simulation** workspace: ■ Tool table ■ Turning tool table ■ Datum table ■ Preset table ■ Grinding tool table ■ Dressing tool table **Further information:** "Tool tables", Page 1995

Action bar



Simulation workspace in the **Editor** operating mode

In the **Editor** operating mode you can test NC programs by simulating them. The simulation helps to detect programming errors or collisions and to check the machining result visually.

The control shows the active tool and the machining time above the action bar.

Further information: "Display of the program run time", Page 187

The action bar contains the following symbols:

Symbol	Function
○	**Control-in-operation**: The control uses the **Control-in-operation** symbol to show the current simulation status in the action bar and on the tab of the NC program: ■ White: no movement command ■ Green: active machining, axes are moving ■ Orange: NC program interrupted ■ Red: NC program stopped
	Simulation speed **Further information:** "Simulation speed", Page 1553
⟲	Reset Return to the beginning of the program, reset transformations and the machining time
▶	Start
▶|	Start in Single Block mode
▶▶|	Run the simulation up to a certain NC block **Further information:** "Simulating an NC program up to a certain NC block", Page 1554

Simulation of tools

The control visualizes the following entries of the tool table in the simulation:

- **L**
- **LCUTS**
- **LU**
- **RN**
- **T-ANGLE**
- **R**
- **R2**
- **KINEMATIC**
- **R_TIP**
- Delta values from the tool table

  Delta values from the tool table increase or decrease the size of the simulated tool. Delta values from the tool call shift the tool in the simulation.

  Further information: "Tool compensation for tool length and tool radius", Page 1112

  Further information: "Tool table tool.t", Page 1995

The control visualizes the following entries of the turning tool table in the simulation:

- **ZL**
- **XL**
- **YL**
- **RS**
- **T-ANGLE**
- **P-ANGLE**
- **CUTLENGTH**
- **CUTWIDTH**

If the **ZL** and **XL** columns are defined in the turning tool table, the indexable insert is displayed and the base body is shown schematically.

Further information: "Turning tool table toolturn.trn (option 50)", Page 2006

The control visualizes the following entries of the grinding tool table in the simulation:

- **R-OVR**
- **LO**
- **B**
- **R_SHAFT**

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010

The control displays the tool in the following colors:

- Turquoise: tool length
- Red: length of cutting edge and tool is engaged
- Blue: length of cutting edge and tool is retracted

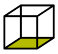
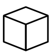
29.2 Pre-defined views

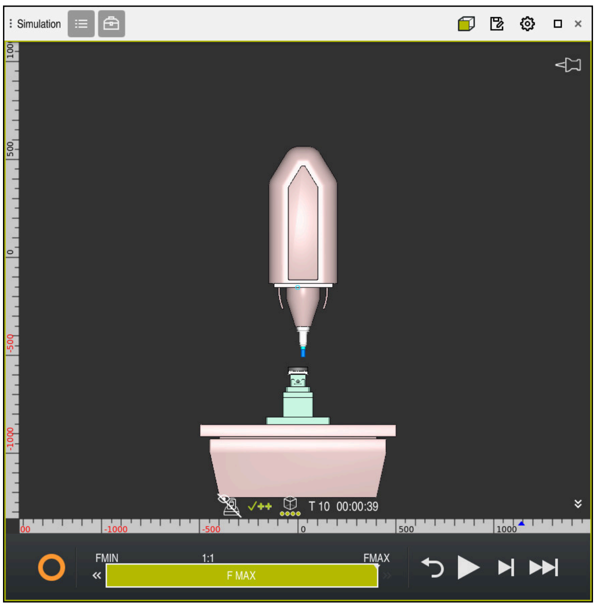
Application

In the **Simulation** workspace, you can choose between various pre-defined views in order to align the workpiece. This allows you to position the workpiece more quickly for the simulation.

Description of function

The control provides the following pre-defined views:

Symbol	Function
	Plan view
	Bottom view
	Front view
	Back view
	Side view (left side)
	Side view (right side)
	Isometric view



Front view of the simulated workpiece in **Machine** mode

29.3 Exporting a simulated workpiece as STL file

Application

In the simulation you can use the **Save** function to save the current status of the simulated workpiece as a 3D model in STL format.

The file size of the 3D model depends on the complexity of the geometry and the selected model quality.

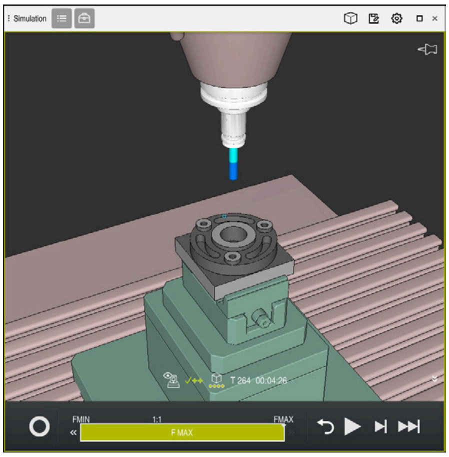
Related topics

- Using an STL file as workpiece blank

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 264
- Modifying an STL file in the **CAD-Viewer** (option 152)

  Further information: "Generating STL files with 3D mesh (option 152)", Page 1472

Description of function



Simulated workpiece

This function can be used only in the **Editor** mode.

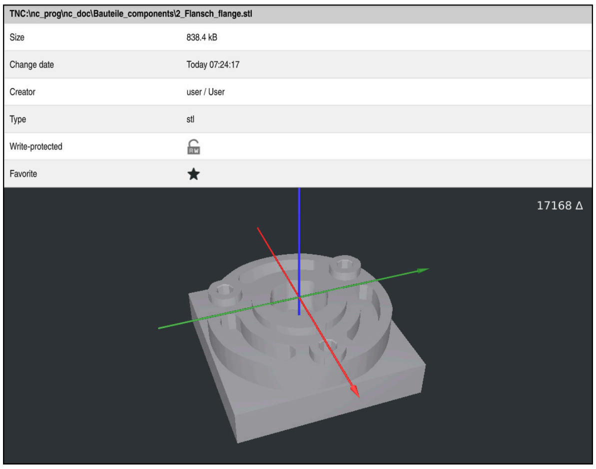
The control can only display STL files with up to 30,000 triangles. If the exported 3D model has too many triangles, due to the excessively high model quality, then you cannot use the exported 3D model on the control.

In this case, reduce the model quality in the simulation.

Further information: "Simulation settings window", Page 1541

You can also use the **3D mesh** function to reduce the number of triangles (option 152).

Further information: "Generating STL files with 3D mesh (option 152)", Page 1472



Simulated workpiece as saved STL file

29.3.1 Saving a simulated workpiece as STL file

To save a simulated workpiece as an STL file:

- Simulate workpiece

- Select **Save**
- > The control opens the **Save as** window.
- Enter the desired file name
- Select **Create**
- > The control saves the created STL file.

29.4 Measuring function

Application

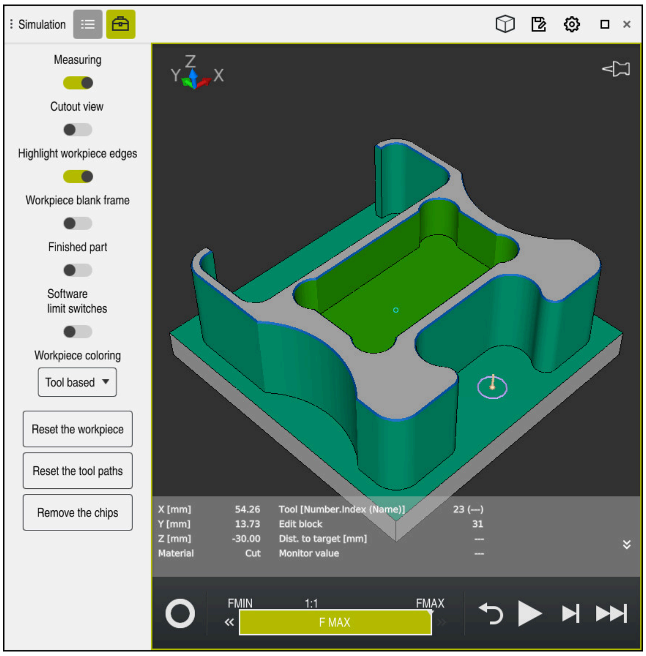
Use the measuring function to measure any points on the simulated workpiece. The control shows various pieces of information about the measured surface.

Requirement

- **Workpiece** mode

Description of function

If you measure a point on the simulated workpiece, the cursor always locks onto the currently selected surface.



Measured point on simulated workpiece

The control shows the following information about the measured surface:

- Measured positions in the **X**, **Y**, and **Z** axes
- Status of the machined surface
  - **Material Cut** = Surface that has been machined
  - **Material NoCut** = Surface that has not been machined
- Cutting tool
- NC block currently running in the NC program
- Distance between the measured surface and the finished part
- Relevant values of monitored machine components (option 155)

  Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1230

29.4.1 Measuring the difference between the workpiece blank and the finished part

To measure the difference between the workpiece blank and the finished part:

- Select an operating mode (e.g., **Editor**)
- Open an NC program with a workpiece blank and finished part defined in **BLK FORM FILE**
- Open the **Simulation** workspace
- Select the **Tool options** column
- Activate the **Measuring** toggle switch
- Select the **Workpiece coloring** selection menu

- Select **Model comparison**
- > The control displays the workpiece blank and finished part defined in the **BLK FORM FILE** function.
- Start the simulation
- > The control simulates the workpiece.
- Select the desired point on the simulated workpiece
- > The control displays the difference in the dimension between the simulated workpiece and the finished part.

> The control uses the **Model comparison** function to identify dimensional differences between the simulated workpiece and the finished part first in color, starting with differences greater than 0.2 mm.

Notes

- If you need to compensate for tools, you can use the measuring function to determine the tool to be compensated for.
- If you notice an error in the simulated workpiece, you can use the measuring function to determine the NC block that causes the error.

29.5 Cutout view in the simulation

Application

In the Cutout view you can cut through the simulated workpiece along any axis. This enables you to check holes and undercuts in the simulation, for example.

Requirement

- **Workpiece** mode

Description of function

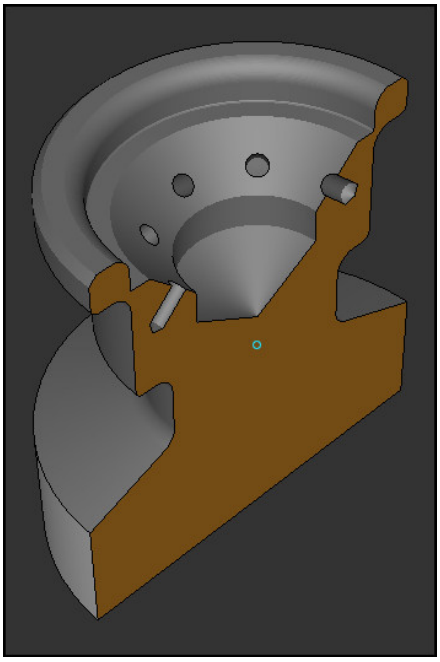
The Cutout view can be used only in the **Editor** mode.

The position of the sectional plane is shown as a percent value when it is shifted in the simulation. The sectional plane is retained until the control is restarted.

29.5.1 Shifting the sectional plane

To shift the sectional plane:

- Select the **Editor** operating mode
- Open the **Simulation** workspace
- Select the **Visualization options** column
- Select **Workpiece** mode
- > The control shows the workpiece view.
- Select the **Workpiece options** column
- Activate the **Cutout view** toggle switch
- > The control activates the **Cutout view**.
- Use the selection menu to choose the desired sectional axis, such as the Z axis
- Use the slider to specify the desired percent value
- > The control simulates the workpiece with the selected sectional settings.



Simulated workpiece in the **Cutout view**

29.6 Model comparison

Application

With the **Model comparison** function you can compare the blank and finished part in STL or M3D format with each other.

Related topics

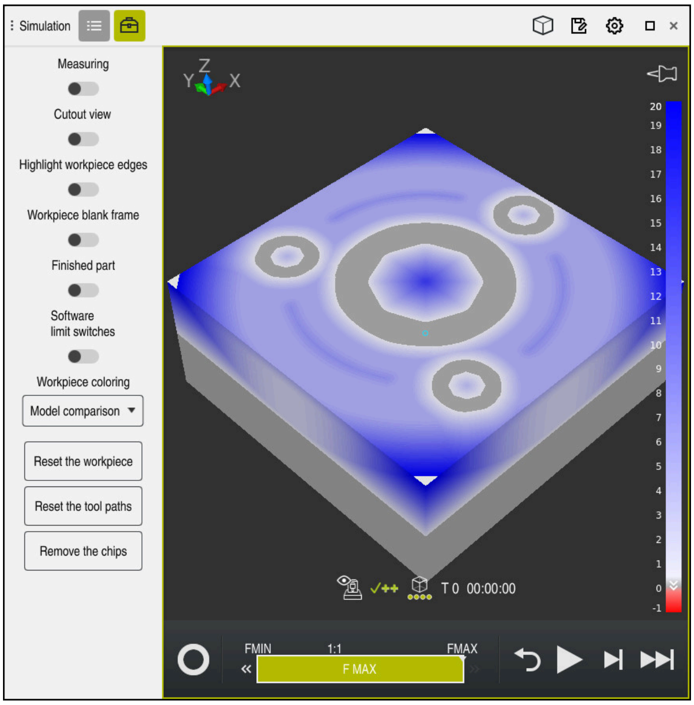
- Programming the blank and finished part with STL files

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 264

Requirements

- STL file or M3D file of workpiece blank and finished part
- **Workpiece** mode
- Workpiece blank definition with **BLK FORM FILE**

Description of function



The control uses the **Model comparison** function to show the difference in material between the models being compared. The control uses a color transition from white to blue to show the difference in material. The more material there is covering the finished part model, the deeper the blue is. When material is removed from the finished part model, the control displays this removal in red.

Notes

- The control uses the **Model comparison** function to identify dimensional differences between the simulated workpiece and the finished part, starting with differences greater than 0.2 mm.
- Use the measuring function to measure the exact dimensional difference between the workpiece blank and the finished part.

  Further information: "Measuring the difference between the workpiece blank and the finished part", Page 1549

29.7 Center of rotation in the simulation

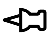
Application

By default, the center of rotation in the simulation is at the center of the model. When you zoom in, the center of rotation is always shifted to the center of the model. If you want to rotate the simulation around a specific point, then you can define the center of rotation manually.

Description of function

Use the **Center of rotation** function to manually set the center of rotation for the simulation.

The control shows the **Center of rotation** symbol as follows, depending on the status:

Symbol	Function
	The center of rotation is at the center of the model.
	The symbol blinks. The center of rotation can be shifted.
	The center of rotation was set manually.

29.7.1 Setting the center of rotation to a corner of the simulated workpiece

To set the center of rotation to a corner of the workpiece:

- Select an operating mode (e.g., **Editor**)
- Open the **Simulation** workspace
- > The center of rotation is at the center of the model.

- Select **Center of rotation**
- > The control switches the **Center of rotation** symbol. The symbol blinks.
- Select a corner of the simulated workpiece
- > The center of rotation is defined. The control switches the **Center of rotation** symbol to "set".

29.8 Simulation speed

Application

You can use a slider to select any speed for the simulation.

Description of function

This function can be used only in the **Editor** operating mode.

The standard speed for the simulation is set to **FMAX**. If you change the simulation speed, then this change is retained until the control is restarted.

You can change simulation speed before as well as during the simulation.

The control provides the following options:

Button	Functions
FMIN	Activate minimum feed rate (**0.01*T**)
«	Reduce the feed rate
1:1	Feed-rate at 1:1 (real-time)
»	Increase the feed rate
FMAX	Activate maximum feed rate (**FMAX**)

29.9 Simulating an NC program up to a certain NC block

Application

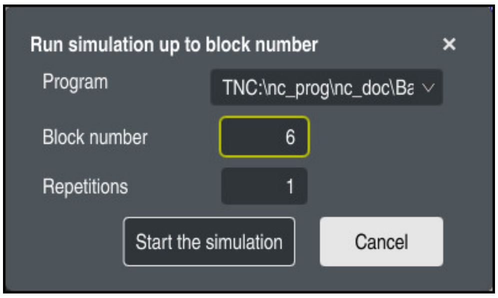
If you want to check a critical point in the NC program then you can simulate the NC program up to a specific NC block that you specify. Once the NC block is reached in the simulation, the control stops the simulation automatically. Starting from this NC block you can then continue the simulation, for example in **Single Block** mode or at a lower simulation speed.

Related topics

- Possibilities in the action bar
  Further information: "Action bar", Page 1543
- Simulation speed
  Further information: "Simulation speed", Page 1553

Description of function

This function can be used only in the **Editor** operating mode.



The **Run simulation up to block number** window with a defined NC block

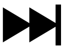
The following settings options are offered in the **Run simulation up to block number** window:

- **Program**
  This field offers a selection menu in which you can choose to simulate up to a specific NC block in the active main program or in a called program.
- **Block number**
  In the **Block number** field, you enter the number of the NC block up to which the simulation should run. The number of the NC block refers to the NC program selected in the **Program** field.
- **Repetitions**
  Use this field if the desired NC block is located within a program-section repeat. Enter in this field up to which iteration of the program-section repeat the simulation should run.
  If you enter **1** or **0** in the **Repetitions** field, the control simulates up to the first iteration of the program section (repetition "0").
  Further information: "Program-section repeats", Page 387

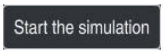
29.9.1 Simulating an NC program up to a certain NC block

To simulate up to a specific NC block:

- Open the **Simulation** workspace



- Select **Run simulation up to block number**
- > The control opens the **Run simulation up to block number** window.
- Use the selection menu in the **Program** field to specify the main program or called program
- Enter the number of the desired NC block in the **Block number** field
- If the block involves a program-section repeat, enter the number of the iteration of the program-section repeat in the **Repetitions** field



- Select **Start the simulation**
- > The control simulates the workpiece up to the selected NC block.

30

Touch Probe Functions in the Manual Operating Mode

30.1 Fundamentals

Application

The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Related topics

- Automatic touch probe cycles
  Further information: "Programmable Touch Probe Cycles", Page 1589
- Preset table
  Further information: "Preset table", Page 2035
- Datum table
  Further information: "Datum table", Page 2045
- Reference systems
  Further information: "Reference systems", Page 1010
- Preassigned variables
  Further information: "Preassigned Q parameters", Page 1368

Requirements

- Calibrated workpiece touch probe
  Further information: "Calibrating the workpiece touch probe", Page 1572

Description of function

The control provides the following functions for setting up the machine in the **Setup** application of the **Manual** operating mode:

- Define the workpiece preset
- Determine and compensate for workpiece misalignment
- Calibrate the workpiece touch probe
- Calibrate the tool touch probe
- Measure the tool

Within the functions, the control provides the following probing methods:

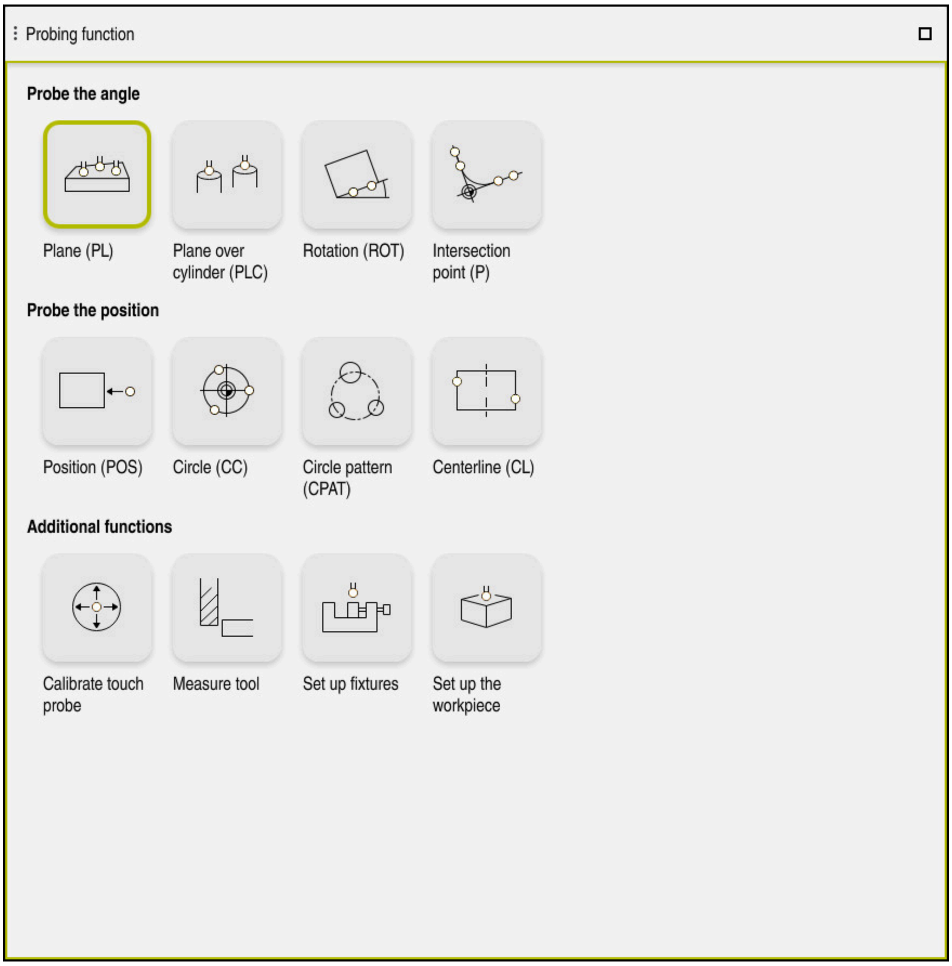
- Manual probing method

  You position and start individual probing processes manually within a touch probe function.

  Further information: "Setting a preset in a linear axis", Page 1565
- Automatic probing method

  You manually position the touch probe to the first probing point before the start of the probing routine and fill out a form with the individual parameters for the respective touch probe function. When you start the touch probe function, the control automatically positions and automatically performs probing.

  Further information: "Determining the circle center point of a stud using the automatic probing method ", Page 1567



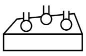
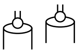
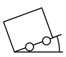
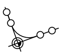
Probing function workspace

Overview

The touch probe functions are structured in the following groups:

Probe the angle

The **Probe the angle** group contains the following touch probe functions:

Button	Function
Plane (PL) 	Use the **Plane (PL)** function to determine the solid angle of a plane. You then save the values in the preset table or align the plane.
Plane over cylinder (PLC) 	Use the **Plane over cylinder (PLC)** function to probe one or two cylinders, each at two different heights. The control calculates the solid angle of a plane from the points probed. You then save the values in the preset table or align the plane.
Rotation (ROT) 	Use the **Rotation (ROT)** function to determine the skew of a workpiece using a straight line. Then save the determined skew as a basic transformation or offset in the preset table. **Further information:** "Determining and compensating the rotation of a workpiece", Page 1569
Intersection point (P) 	Use the **Intersection point (P)** function to probe four probing objects. The probing objects can be either positions or circles. The control determines the intersection of the axes and the skew of the workpiece from the objects that have been probed. You can set the intersection point as a preset. You can transfer the determined skew to the preset table as a basic transformation or as an offset.

The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.

Further information: "Preset table", Page 2035

You can compensate for the workpiece misalignment by rotating the table only if the machine is designed with a rotary table axis that is oriented perpendicularly with respect to the workpiece coordinate system **W-CS**.

Further information: "Comparison of offset and 3D basic rotation", Page 1580

Probe the position

The **Probe the position** group contains the following touch probe functions:

Button	Function
Position (POS)	You can use the **Position (POS)** function to probe a position in the X axis, Y axis or Z axis. **Further information:** "Setting a preset in a linear axis", Page 1565
Circle (CC)	The **Circle (CC)** function is used to determine the coordinates of a circle center point (e.g., for a hole or for a stud). **Further information:** "Determining the circle center point of a stud using the automatic probing method ", Page 1567
Circle pattern (CPAT)	The **Circle pattern (CPAT)** function is used to determine the center point coordinates of a circle pattern.
Centerline (CL)	The **Centerline (CL)** function is used to determine the center point of a ridge or slot.

Additional functions group

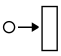
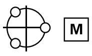
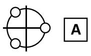
The **Additional functions** group contains the following touch probe functions:

Button	Function
Calibrate touch probe	The **Calibrate touch probe** function is used to determine the length and radius of a workpiece touch probe. **Further information:** "Calibrating the workpiece touch probe", Page 1572
Measure tool	The **Measure tool** function allows you to measure tools by scratching. In this function, the control supports milling tools, drilling tools and turning tools.
Set up fixtures	The **Set up fixtures** function is used to determine the position of a fixture in the machine's working space using a workpiece touch probe (option 140). **Further information:** "Integrating the fixtures into collision monitoring (option 140)", Page 1174
Set up the workpiece	The **Set up the workpiece** function is used to determine the position of a workpiece in the machine's working space using a workpiece touch probe (option 159). **Further information:** "Setting up the workpiece with graphical support (option 159)", Page 1582

Buttons

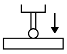
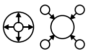
General buttons in the touch probe functions

The following buttons are available, depending on the selected touch probe function:

Button	Function
	Finish the active touch probe function
	Select the workpiece preset and the pallet presets and edit values if required **Further information:** "Change the preset window", Page 1564 **Further information:** "Preset table", Page 2035 The control grays out this icon while a probing process takes place. In this condition, you can check the presets but you cannot edit them. You need to stop the probing process in order to edit presets.
	Display help graphics for the selected touch probe function
Y+	Select the probing direction
	Apply the actual position
	Approaching and probing points on a straight surface manually
	Approaching and probing points on a stud or in a hole manually
	Approaching and probing points on a stud or in a hole automatically After the last touching process and if the opening angle contains the value 360°, the control positions the workpiece touch probe back to the position it had prior to starting the probing function.

Calibration buttons

The control offers the following functions for calibrating a 3D touch probe:

Button	Function
	Calibrating the length of a 3D touch probe
	Calibrating the radius of a 3D touch probe
Apply calibration data	Transferring values from the calibration process into tool management

Further information: "Calibrating the workpiece touch probe", Page 1572

You can calibrate a 3D touch probe by using a calibration standard, such as a calibrating ring.

The control provides the following options:

Button	Function
	Measure the radius and the center offset using a calibration ring
	Measure the radius and the center offset using a stud or a calibration pin
	Measure the radius and the center offset using a calibration sphere Optionally calibrating the tool touch probe in 3D (option 92) **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1140 **Further information:** "3D calibration (option 92)", Page 1573

Buttons in the Working plane is inconsistent! window

If the positions of the rotary axes do not match the tilting situation in the **3-D rotation** window, the control opens the **Working plane is inconsistent!** window.

The control offers the following functions in the **Working plane is inconsistent!** window:

Button	Function
3-D ROT Apply status	The **3-D ROT Apply status** function transfers the position of the rotary axes into the **3-D rotation** window. **Further information:** "3-D rotation window (option 8)", Page 1098
3-D ROT Ignore status	The **3-D ROT Ignore status** function makes the control calculate the probing results, assuming that the rotary axes are in their zero position.
Align the rotary axes	The **Align the rotary axes** function aligns the rotary axes to the active tilting situation in the **3-D rotation** window.

Buttons for measured values

After executing a touch probe function, you select the desired control reaction.

The control offers the following functions:

Button	Function
Compensate the active preset	The **Compensate the active preset** function transfers the measuring result into the active line of the preset table. **Further information:** "Preset table", Page 2035
Write the datum	The **Write the datum** function transfers the measuring result into a desired line of the datum table. **Further information:** "Datum table", Page 2045
Align rotary table	The **Align rotary table** function aligns the rotary axes mechanically according to the measuring result.

Change the preset window

In the **Change the preset** window you can select a preset or edit the values of a preset.

Further information: "Preset management", Page 1025

The **Change the preset** window provides the following buttons:

Button	Meaning
Reset basic rotation	The control resets the values from the columns **SPA**, **SPB** and **SPC**.
Reset offsets	The control resets the values from the columns **A_OFFS**, **B_OFFS** and **C_OFFS**.
Apply	The control saves the changes and the selected preset. Then the control closes the window.
Reset	The control cancels the changes and restores the initial condition.
Cancel	The control closes the window without saving.

If you change a value, the control marks this value with a blue dot.

Log file of touch probe cycles

After executing the respective touch-probe cycle, the control writes the measured values to the TCHPRMAN.html file.

You can check the readings of past measurements in the **TCHPRMAN.html** file.

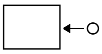
If you have not defined a path in the machine parameter **FN16DefaultPath** (no. 102202), the control will store the TCHPRMAN.html file directly under **TNC:**.

If you run several touch probes cycles in a row, the control stores the measured values below each other.

30.1.1 Setting a preset in a linear axis

To probe the preset in any axis:

- Select the **Manual** operating mode
- Call the workpiece touch probe as a tool
- Select the **Setup** application

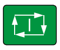


- Select the **Position (POS)** touch probe function
- > The control opens the **Position (POS)** touch probe function.

- Select **Change the preset**
- > The control opens the **Change the preset** window.
- Select the desired row of the preset table
- > The control highlights the selected line in green.

- Press **Apply**
- > The control activates the selected line as the workpiece preset.
- Use the axis keys to position the workpiece touch probe at the desired probing position (e.g., above the workpiece in the workspace)

- Select the probing direction (e.g., **Z-**)



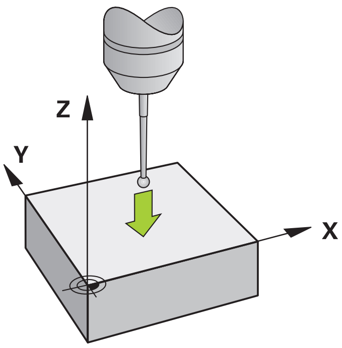
- Press the **NC start** key
- > The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- > The control shows the measurement results.
- In the **Nominal value** area, enter the new preset of the probed axis (e.g., **1**)

Compensate the active preset

- Select **Compensate the active preset**
- > The control enters the defined nominal value in the preset table.
- > The control marks the row with an icon.

> If you use the **Write the datum** function, the control also marks this row with an icon.
>
> When you have completed the probing process in the first axis, you can probe up to two additional axes using the **Position (POS)** probing function.

- Select **Exit probing**
- > The control closes the **Position (POS)** probing function.



30.1.2 Determining the circle center point of a stud using the automatic probing method

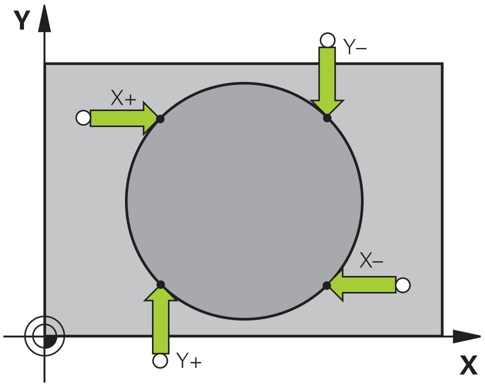
To probe a circle center point:

- Select the **Manual** operating mode
- Call the workpiece touch probe as a tool
  Further information: "Manual operation application", Page 202
- Select the **Setup** application
- Select **Circle (CC)**
- > The control opens the **Circle (CC)** probing function.
- If necessary, select another preset for the probing process
- Select measuring method **A**
- Select **Type of contour** (e.g., stud)
- Enter **Diameter** (e.g., 60 mm)
- Enter **Starting angle** (e.g., –180°)
- Enter **Angular length** (e.g., 360°)
- Position the 3D touch probe at the desired probing position next to the workpiece and below the workpiece surface
- Select the probing direction (e.g., **X+**)
- Turn the feed rate potentiometer to zero
- Press the **NC start** key
- Slowly turn on the feed rate potentiometer
- > The control executes the touch probe function based on the data entered.
- > The control shows the measurement results.
- In the **Nominal value** area, enter the new preset of the scanned axes (e.g., **0**)

- Select **Compensate the active preset**
- > The control sets the preset to the entered nominal value.
- > The control marks the row with an icon.

> If you use the **Write the datum** function, the control also marks this row with an icon.

- Select **Exit probing**
- > The control closes the **Circle (CC)** probing function.



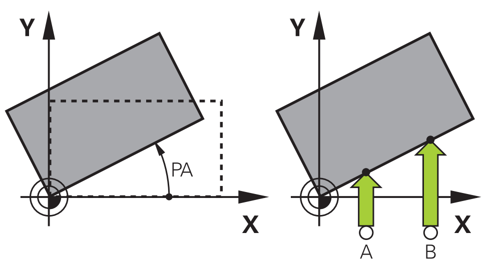
30.1.3 Determining and compensating the rotation of a workpiece

To probe the rotation of a workpiece:

- Select the **Manual** operating mode
- Call the 3D touch probe as a tool
- Select the **Setup** application
- Select **Rotation (ROT)**
- > The control opens the **Rotation (ROT)** probing function.
- If necessary, select another preset for the probing process
- Position the 3D touch probe at the desired probing position in the workspace
- Y+ Select the probing direction (e.g., **Y+**)
- Press the **NC start** key
- > The control executes the first probing process and limits the subsequently selectable probing directions.
- Position the 3D touch probe at the second probing position in the workspace
- Press the **NC start** key
- > The control executes the probing process and then shows the measurement results.
- Compensate the active preset — Select **Compensate the active preset**
- > The control transfers the determined basic rotation to the **SPC** column of the active line of the preset table.
- > The control marks the row with an icon.

> Depending on the tool axis, the measurement result can also be written to another column of the preset table (e.g., **SPA**).

- Select **Exit probing**
- > The control closes the **Rotation (ROT)** probing function.



30.1.4 Using touch probe functions with mechanical probes or dial gages

If your machine does not have an electronic 3D touch probe, you can use all manual touch probe functions with manual probing methods with mechanical buttons or with scratching.

For this, the control provides the **Accept position** button.

To determine a basic rotation with a mechanical probe:

- ▶ Select the **Manual** operating mode
- ▶ Insert the tool, such as an analog 3D probe or feeler lever gage
- ▶ Select the **Setup** application
- ▶ Select the **Rotation (ROT)** probing function
- ▶ Select the probing direction (e.g., **Y+**)
- ▶ Move the mechanical probe to the first position to be captured by the control.
- ▶ Select **Accept position**
- > The control saves the current position.
- ▶ Move the mechanical probe to the next position to be captured by the control.
- ▶ Select **Accept position**
- > The control saves the current position.
- ▶ Select **Compensate the active preset**
- > The control transfers the determined basic rotation to the active line of the preset table.
- > The control marks the row with an icon.

> The determined angles have different effects depending on whether they are transferred as an offset or as a basic rotation to the corresponding table.
>
> **Further information:** "Comparison of offset and 3D basic rotation", Page 1580

- ▶ Select **Exit probing**
- > The control closes the **Rotation (ROT)** probing function.

Notes

- When you use a non-contacting tool touch probe, use touch probe, then you are using touch probe functions from the third-party manufacturer. This is the case, for example, with a laser touch probe. Refer to your machine manual.
- The accessibility of the pallet preset table in the touch probe functions depends on the machine manufacturer's configuration. Refer to your machine manual.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) temporarily.

  Further information: "Global Program Settings (GPS, option 44)", Page 1217
- You can use the manual touch probe functions only with restrictions in turning mode (option 50).
- You must calibrate the touch probe separately in turning mode. The factory default setting of the worktable may vary between milling mode and turning mode, which is why you must calibrate the touch probe without any center offset in turning mode. You can create a tool index for storing the additionally calibrated tool data in the same tool.

  Further information: "Indexed tool", Page 276
- When probing while the guard door is open and spindle orientation to probing direction is active, the number of spindle revolutions is limited. When the maximum permitted number of spindle revolutions is reached, the direction of spindle rotation changes and the control may no longer orient the spindle on the shortest path.
- If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine manufacturer has defined.
- When writing into an empty line of the preset table, the control automatically fills the other columns with values. To define a preset completely, you must determine the values in all axes and write them into the preset table.
- If no tool touch probe is inserted, the actual position can be captured with **NC START**. The control displays a warning that no probing movement is carried out in that case.
- Recalibrate the workpiece touch probe in the cases below:
  - Initial configuration
  - Broken stylus
  - Stylus replacement
  - Change in the probe feed rate
  - Irregularities caused, for example, when the machine heats up
  - Change of active tool axis

Definition

Spindle tracking

If the **Track** parameter in the touch probe table is active, the control orients the workpiece probing system so that the same position is always used for probing. By deflecting in the same direction, you can reduce the measurement error to the repeatability of the workpiece probing system. This behavior is called spindle tracking.

30.2 Calibrating the workpiece touch probe

Application

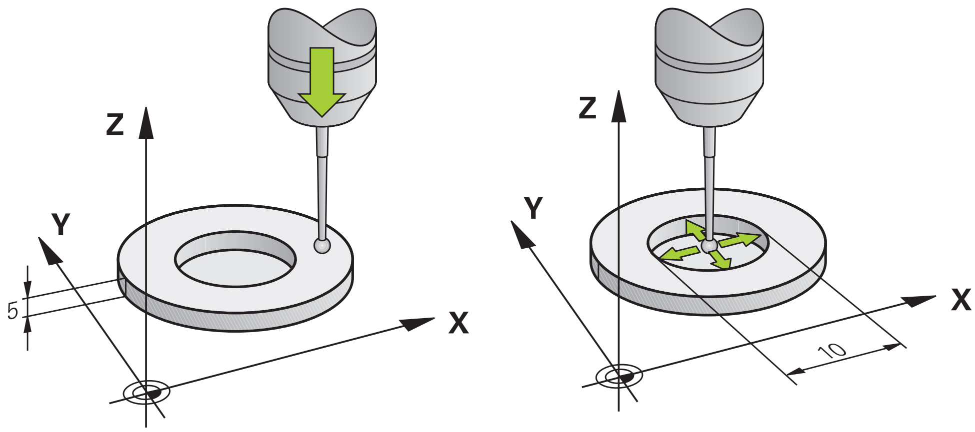
In order to precisely specify the actual trigger point of a 3D touch probe, you must first calibrate the touch probe, otherwise the control cannot provide precise measuring results.

During 3D calibration, you determine the angle-dependent deflection behavior of a workpiece touch probe in any probing direction (option 92).

Related topics

- Calibrate the workpiece touch probe automatically
  Further information: "Touch Probe Cycles: Calibration", Page 1848
- Touch probe table
  Further information: "Touch probe table tchprobe.tp", Page 2022
- 3D radius compensation depending on the contact angle (option 92)
  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1140

Description of function



During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 271

You can calibrate the workpiece touch probe with various tools. For example, the workpiece touch probe can be calibrated using an overmilled surface in length and a calibration ring in the radius. This creates a reference between the workpiece touch probe and the tools in the spindle. In this procedure, measured tools and the calibrated workpiece touch probe correspond using the tool presetting device.

Calibrating an L-shaped stylus

Before you calibrate an L-shaped stylus you first must define the parameters in the touch probe table. Based on these approximate values, the control can align the touch probe during the calibration and determine the actual values.

At first, define the following parameters in the touch probe table:

Parameter	Value to be defined
CAL_OF1	Length of extension The extension is the angled length of the L-shaped stylus.
CAL_OF2	0
CAL_ANG	Spindle angle at which the extension is parallel to the main axis For this, manually position the extension in the direction of the main axis and read the value from the position display.

After the calibration, the control overwrites the previously defined values in the touch probe table with the determined values.

Further information: "Touch probe table tchprobe.tp", Page 2022

When calibrating the length, the control aligns the touch probe with the calibration angle defined in the **CAL_ANG** column.

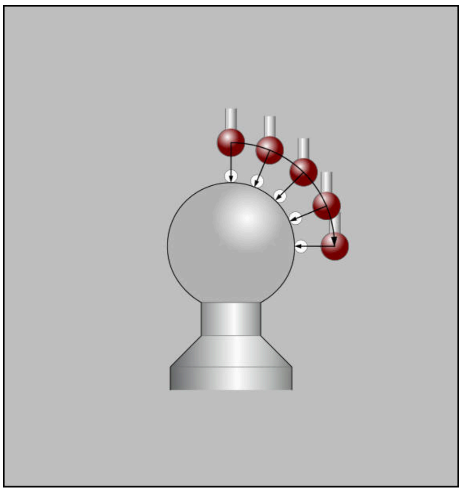
While calibrating the touch probe, ensure that the feed rate override is 100%. That way you can always use the same feed rate for the subsequent probing processes as was used for the calibration. Hence, you can exclude inaccuracies during the probing caused by modified feed rates.

3D calibration (option 92)

In addition to calibrating with a calibration sphere, the control also enables the touch probe to be calibrated dependent on the angle. For this purpose the control probes the calibration sphere in a quarter circle in the perpendicular. The 3D calibration data specifies the deflection behavior of the touch probe in any probing direction.

The control saves the deviations in a compensation value table ***.3DTC** in the folder **TNC:\system\3D-ToolComp**.

The control creates a specific table for each calibrated touch probe. In the tool table the **DR2TABLE** column is automatically referenced to this.



3D calibration

Reversal measurement

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

HEIDENHAIN touch probes are predefined as to whether or how a touch probe can be oriented. Other touch probes are configured by the machine manufacturer.

When calibrating the radius, up to three circular measurements can be taken depending on the possible orientation of the workpiece touch probe. The first two circular measurements determine the center offset of the workpiece touch probe. The third circular measurement determines the effective stylus tip radius. If orientation of the spindle is not possible or only a certain orientation is possible due to the workpiece touch probe, circular measurements are omitted.

30.2.1 Calibrating the length of the workpiece touch probe

To calibrate a workpiece touch probe using an overmilled surface in length:

- Measure the end milling cutter on the tool presetting device
- Store the measured end milling cutter in the tool magazine of the machine
- Enter the tool data of the end milling cutter in tool management
- Clamp the workpiece blank
- Select the **Manual** operating mode
- Replace the end milling cutter in the machine
- Switch on spindle (e.g., with **M3**)
- Use the handwheel to scratch the workpiece blank

  Further information: "Setting a preset with milling cutters", Page 1026
- Set preset in the tool axis (e.g., with **Z**)
- Position the end milling cutter next to the workpiece blank
- Set a small value in the tool axis (e.g., with **-0.5 mm**)
- Overmill the workpiece blank using the handwheel
- Set the preset again in the tool axis (e.g., with **Z=0**)
- Switch off spindle (e.g., with **M5**)
- Replace the tool touch probe
- Select the **Setup** application
- Select **Calibrate touch probe**
- Select the **Length calibration** measurement method
- > The control displays the current calibration data.
- Enter the reference surface position (e.g., with **0**)
- Position the workpiece touch probe close to the surface of the overmilled area

> Check that the area to be probed is flat and free of chips before you start the touch probe function.

- Press the **NC start** key
- > The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- Check results
- Select **Apply calibration data**
- > The control transfers the calibrated length of the 3D touch probe to the tool table.
- Select **Exit probing**
- > The control closes the **Calibrate touch probe** function.

30.2.2 Calibrating the radius of the workpiece touch probe

To calibrate a workpiece touch probe using a setting ring in the radius:

- Clamp the setting ring on the machine table (e.g., with clamps)
- Select the **Manual** operating mode
- Position the 3D touch probe in the hole of the setting ring

> Make sure that the stylus tip is completely recessed into the calibration ring. This causes the control to probe with the largest point of the stylus tip.

- Select the **Setup** application
- Select **Calibrate touch probe**
- Select **Radius** measurement method
- Select **Setting ring** calibration standard
- Enter the diameter of the ring gauge
- Enter the start angle
- Enter the number of touch points
- Press the **NC Start** key
- > The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results
- Select **Apply calibration data**
- > The control stores the calibrated radius of the 3D touch probe in the tool table.
- Select **Exit probing**
- > The control closes the **Calibrate touch probe** function.

30.2.3 3D calibration of workpiece touch probe (option 92)

To calibrate a workpiece touch probe using a calibration sphere in the radius:

- Clamp the setting ring on the machine table (e.g., with clamps)
- Select the **Manual** operating mode
- Position the workpiece touch probe centrally above the sphere
- Select the **Setup** application
- Select **Calibrate touch probe**
- Select **Radius** measurement method
- Select the **Calibration sphere** calibration standard
- Enter the diameter of the sphere
- Enter the start angle
- Enter the number of touch points
- Press the **NC Start** key
- The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results
- Select **Apply calibration data**
- The control stores the calibrated radius of the 3D touch probe in the tool table.
- The control shows the **3D calibration** measurement method.
- Select the **3D calibration** measurement method
- Enter the number of touch points
- Press the **NC Start** key
- The 3D touch probe probes all required touch points in an automatic probing routine.
- Select **Apply calibration data**
- The control saves the deviations in a compensation value table under **TNC:\system\3D-ToolComp**.
- Select **Exit probing**
- The control closes the **Calibrate touch probe** function.

Instructions for calibration

- In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.
- If you press the **OK** button after the calibration process, the control accepts the calibration values for the active touch probe. The updated tool data then becomes immediately effective, and it is not necessary to repeat the tool call.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- If you want to calibrate using the outside of an object, you need to pre-position the touch probe above the center of the calibration sphere or calibration pin. Ensure that the probing points can be approached without collisions.
- The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset in the touch probe table. The control uses the **TP_NO** parameter to link the data from the touch probe table with the data from the tool table.

  Further information: "Touch probe table tchprobe.tp", Page 2022

30.3 Suppressing touch probe monitoring

Application

If you move a workpiece touch probe too close to the workpiece, you can accidentally deflect the workpiece touch probe. You cannot retract a deflected workpiece touch probe in the monitored state. You can retract a deflected workpiece touch probe by suppressing touch probe monitoring.

Description of function

If the control does not receive a stable signal from the probe, the button displays **Suppress touch probe monitoring**.

As long as touch-probe monitoring is switched off, the control displays the error message **The touch probe monitor is deactivated for 30 seconds**. This error message remains active only for 30 seconds.

30.3.1 Deactivating touch probe monitoring

To deactivate touch probe monitoring:

- ▶ Select the **Manual** operating mode
- ▶ Select **Suppress touch probe monitoring**
- > The control disables touch-probe monitoring for 30 seconds.
- ▶ If required, move the touch probe so that the control receives a stable signal from it.

Notes

NOTICE

Danger of collision!

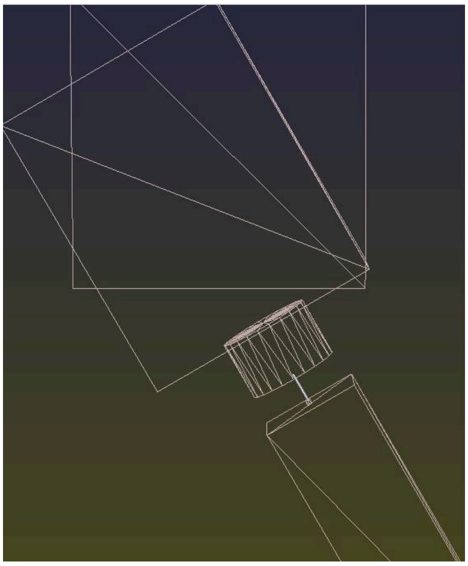
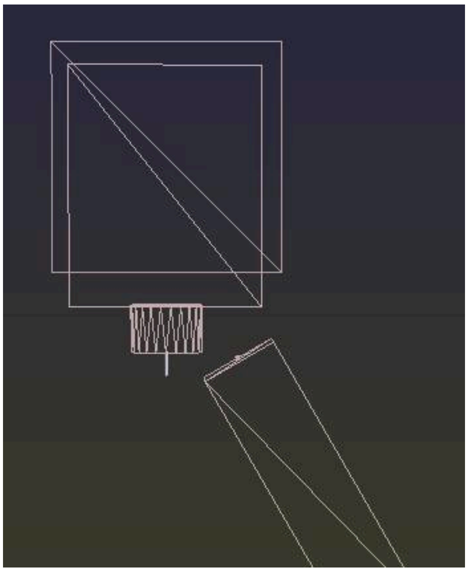
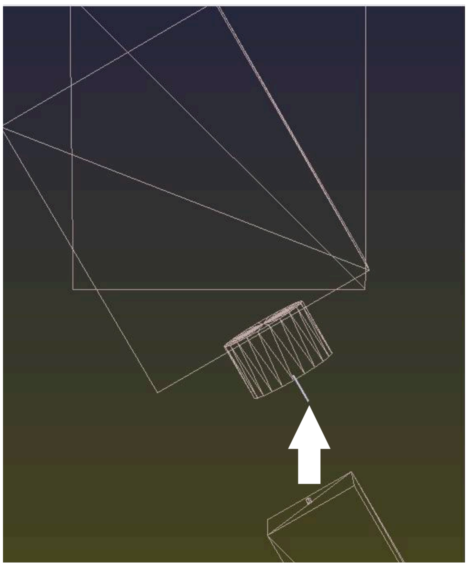
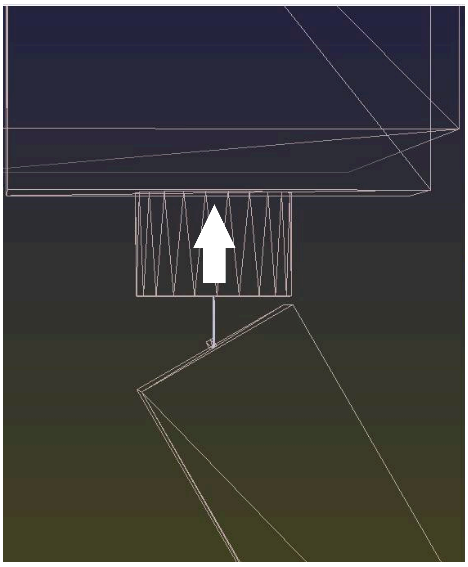
While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse!

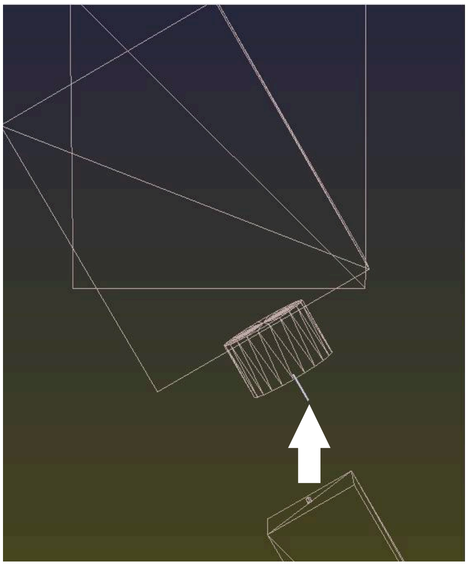
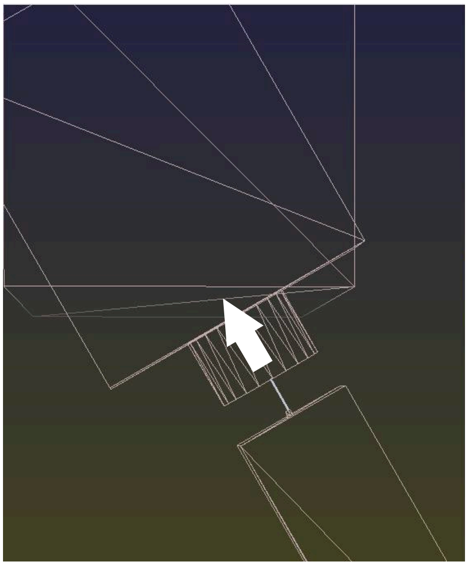
- ▶ Carefully move the axes in the **Manual** operating mode

If the touch probe sends a stable signal within the 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

30.4 Comparison of offset and 3D basic rotation

The following example shows how the two functions differ.

<table>
<tr><th>Offset</th><th>3D basic rotation</th></tr>
<tr><td>Initial state

Position display:
Actual position
B = 0
C = 0
Preset table:
SPB = 0
B_OFFS = -30
C_OFFS = +0</td><td>Initial state

Position display:
Actual position
B = 0
C = 0
Preset table:
SPB = -30
B_OFFS = +0
C_OFFS = +0</td></tr>
<tr><td>Movement in +Z without tilting
</td><td>Movement in +Z without tilting
</td></tr>
</table>

Offset	**3D basic rotation**
Movement in +Z with tilting **PLANE SPATIAL** with **SPA+0 SPB+0 SPC +0**	Movement in +Z with tilting **PLANE SPATIAL** with **SPA+0 SPB+0 SPC +0**
	
> The orientation **is not correct**!	> The orientation is correct! > The next machining step will be **correct**.

HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

30.5 Setting up the workpiece with graphical support (option 159)

Application

Use the **Set up the workpiece** function to determine the position and misalignment of a workpiece with only one touch-probe function and save it as a workpiece preset. Tilting and probing on curved surfaces can be performed during setup in order to probe also complex workpieces, such as free-form parts.

The control supports you additionally by showing the clamping situation and possible touch points in the **Simulation** workspace by means of a 3D model.

Related topics

- Touch-probe functions in the **Setup** application

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557
- Generating an STL file of a workpiece

  Further information: "Exporting a simulated workpiece as STL file", Page 1546
- **Simulation** workspace

  Further information: "Simulation Workspace", Page 1535
- Calibrating fixtures with graphical support (option 140)

  Further information: "Integrating the fixtures into collision monitoring (option 140)", Page 1174

Requirements

- Advanced Functions Set 2 (software option 9)
- Model Aided Setup (software option 159)
- Touch probe properly defined in the tool management:
  - Spherical radius in the **R2** column
  - If probing on inclined surfaces, the spindle tracking in the **TRACK** column needs to be active

  Further information: "Tool data for touch probes", Page 295
- Workpiece touch probe calibrated

  If probing on inclined surfaces, a 3D calibration of the touch probe needs to be performed (option 92).

  Further information: "Calibrating the workpiece touch probe", Page 1572
- 3D model of the workpiece as STL file

  The STL file may contain up to 300,000 triangles. The more the 3D model corresponds to the actual workpiece, the higher the possible workpiece setup accuracy.

  If applicable, optimize the 3D model with the **3D mesh** function (option 152).

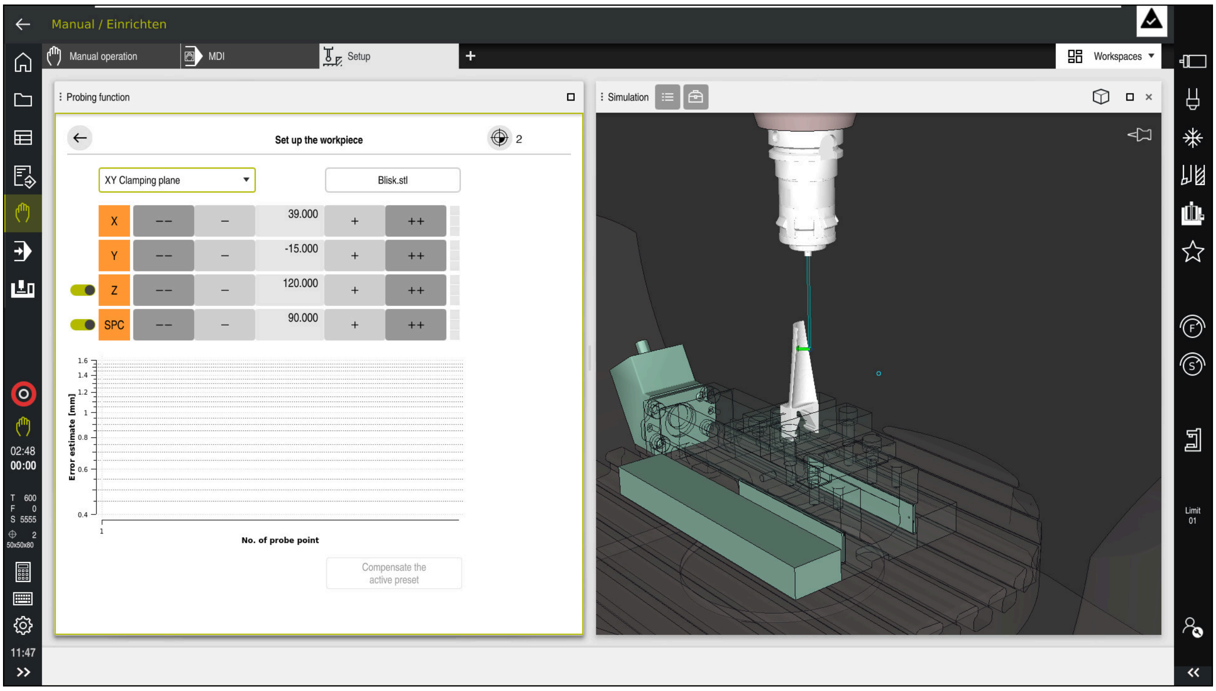
  Further information: "Generating STL files with 3D mesh (option 152)", Page 1472

Description of function

The **Set up the workpiece** function is available as a touch probe function in the **Setup** application of the **Manual** operating mode.

Extensions of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphical support for setting up the workpiece.



Set up the workpiece function with the **Simulation** workspace open

When the **Set up the workpiece** function is active, the **Simulation** workspace displays the content below:

- Current position of workpiece as viewed by the control
- Probed points on the workpiece
- Possible direction of probing by means of an arrow:
  - No arrow

    Probing is not possible. The workpiece touch probe is too distant from the workpiece or the workpiece touch probe is positioned within the workpiece, as seen by the control.

    In this case you can correct the position of the 3D model in the simulation, if required.
  - Red arrow

    Probing in the direction of the arrow is not possible.

    > Probing on edges, corners or heavily curved workpiece areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    Probing in the direction of the arrow is possible to a limited extent. The probing is performed in a deselected direction or could cause collisions.
  - Green arrow

    Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up the workpiece** function contains the following icons and buttons:

Icon or button	Function
	Open the **Change the preset** window You can select the workpiece preset and the pallet preset and edit values if required. ℹ After the first point has been probed, the control grays out the icon.
XY Clamping plane	Use this selection menu to define the probing mode. Depending on the probing mode, the control displays the respective axis directions and spatial angles. **Further information:** "Probing mode", Page 1585
1_plate_blk.stl	File name of 3D model
--	Shifts the position of the virtual workpiece by 10 mm or 10° in the negative axis direction ℹ Shifts the workpiece in mm in a linear axis and in degrees in a rotary axis.
-	Shifts the position of the virtual workpiece by 1 mm or 1° in the negative axis direction
91.349 ± 0.007	■ Enter the position of the virtual workpiece directly ■ Value and estimated accuracy of the value after the probing
+	Shifts the position of the virtual workpiece by 1 mm or 1° in the positive axis direction
++	Shifts the position of the virtual workpiece by 10 mm or 10° in the positive axis direction
	Status of direction The control displays the following colors: ■ Gray The axis direction is deselected in this setup process and is not considered. ■ White No touch points have been determined yet. ■ Red The control cannot locate the workpiece position in this axis direction. ■ Yellow The position of the workpiece in this axis already contains information. The information is not meaningful yet. ■ Green The control can locate the workpiece position in this axis direction.
Compensate the active preset	The control saves the determined values in the active row of the preset table.

Probing mode

The following modes for probing the workpiece are available to you:

- **XY Clamping plane**

  X, **Y** and **Z** axis directions as well as spatial angle **SPC**
- **XZ Clamping plane**

  X, **Y** and **Z** axis directions as well as spatial angle **SPB**
- **YZ Clamping plane**

  X, **Y** and **Z** axis directions as well as spatial angle **SPA**
- **6D**

  X, **Y** and **Z** axis directions as well as spatial angles **SPA**, **SPB** and **SPC**

Depending on the probing mode, the control displays the respective axis directions and spatial angles. In the **XY**, **XZ** and **YZ** clamping planes a toggle switch allows you to deselect the respective tool axis and spatial angle, if required. The control will not take deselected axis directions into account in the setup process and positions the workpiece by considering the remaining axis directions only.

HEIDENHAIN recommends executing the setup process as follows:

1. Pre-position a 3D model in the machine's working space

   At this point in time, the control does not know the precise position of the workpiece, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe produces values close to the position of the real workpiece.
2. Set the first touch points in the **X**, **Y** and **Z** axis directions

   If the control can determine the position in one axis direction, it will change the status of that axis to green.
3. Determine the spatial angle by setting further touch points

   To achieve maximum accuracy when probing the spatial angles, the touch points should be as far apart from one another as possible.
4. Increase the accuracies by additional check points

   Additional check points at the end of the measuring process improve the matching accuracy and minimize the misalignment between the 3D model and the real workpiece. Perform as many probing processes as necessary until the control displays the desired accuracy beneath the current value.

The error estimate diagram shows for each touch point the approximate distance of the 3D model from the real workpiece.

Further information: "Error estimate diagram", Page 1586

Error estimate diagram

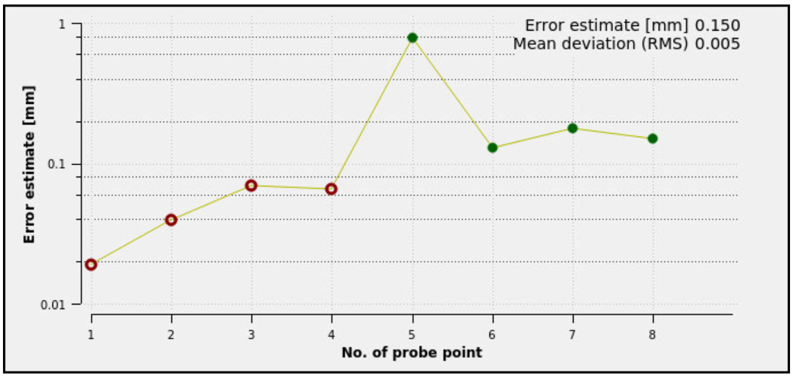
Every additional touch point gradually restricts the possible positioning of the workpiece and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows the estimated value of the distance of the 3D model from the real workpiece. For this purpose, the control considers not only the touch points, but the whole workpiece.

When the error estimate diagram shows green circles and the desired accuracy, the setup process will be complete.

The following factors influence the accuracy that can be achieved when measuring workpieces:

- Accuracy of workpiece touch probe
- Accuracy of the machine kinematic configuration
- Deviations of the 3D model from the real workpiece
- Condition of the actual workpiece (e.g., unmachined areas)



Error estimate diagram in the **Set up the workpiece** function

The error estimate diagram of the **Set up the workpiece** function shows the following information:

- **Mean deviation (RMS)**

  This area shows the average distance of the real workpiece from the 3D model in mm.
- **Error estimate [mm]**

  This axis shows the course of the error estimate based on the individual touch points. The control shows red circles until it can determine all axis directions. From then on the control will show green circles.
- **No. of probe point**

  This axis shows the numbers of the individual probing points.

30.5.1 Setting up the workpiece

Use the **Set up the workpiece** function to set the preset:

- ▶ Affix a real workpiece in the machine's working space
- ▶ Select the **Manual** operating mode
- ▶ Insert the workpiece touch probe
- ▶ Manually position the workpiece touch probe above the workpiece at a notable point (e.g., a corner)

> This step makes the subsequent steps easier.

- ▶ Select the **Setup** application
- ▶ Select **Set up the workpiece**
- \> The control opens the **Set up the workpiece** menu.
- ▶ Select a 3D model matching the real workpiece
- ▶ Select **Open**
- \> The control opens the selected 3D model in the simulation.
- ▶ If necessary, open the **Change the preset** window
- ▶ Select a new preset if necessary
- ▶ Select the **Apply** function if necessary
- ▶ Pre-position the 3D model by using the buttons for the individual axis directions within the virtual working space of the machine

> For pre-positioning the workpiece, use the workpiece touch probe as a point of reference.
>
> Even during the setup process, the shift functions are available for correcting the fixture position manually. Then, probe a new point.

- ▶ Define the probing mode (e.g., **XY Clamping plane**)
- ▶ Position the workpiece touch probe until the control shows a green arrow pointing downward

> As the 3D model is only pre-positioned at this point in time, the green arrow cannot provide any reliable information about whether the desired surface of the workpiece will actually be probed. Check if the workpiece position in the simulation and in the machine match each other and if probing in the direction of the arrow is possible on the machine.
>
> Do not probe directly near edges, chamfers and roundings.

- ▶ Press the **NC start** key
- \> The control probes in the direction of the arrow.
- \> The control displays the status of the **Z** axis in green and shifts the workpiece to the probed position. The control marks the probed position with a point in the simulation.

- Repeat this process in axis directions **X+** and **Y+**
- > The control changes the status of the axes to green.
- Probe another point in axis direction **Y+** for the basic rotation
- > The control changes the status of the **SPC** spatial angle to green.
- Probe the check point in axis direction **X-**

Compensate the active preset

- Select **Compensate the active preset**
- > The control saves the determined values in the active row of the preset table.
- Exit the **Set up the workpiece** function

Notes

NOTICE

Danger of collision!

To probe the clamping situation in the machine exactly, the workpiece touch probe must be properly calibrated and the value **R2** properly defined in the tool management. Otherwise, incorrect tool data of the workpiece touch probe may cause inaccurate measurement and possibly a collision.

- Calibrate the workpiece touch probe at regular intervals
- Enter parameter **R2** in the tool management

- The control cannot identify modeling differences between the 3D model and the workpiece.
- Collisions might be more easily detected, if a tool carrier is assigned to the workpiece touch probe.
- HEIDENHAIN recommends probing check points for one axis direction on both sides of the workpiece. As a result, the control will correct the position of the 3D model in the simulation uniformly.

31

Programmable Touch Probe Cycles

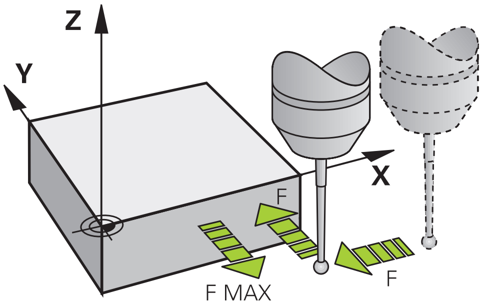
31.1 Working with Touch Probe Cycles

31.1.1 General information about touch probe cycles

Method of function

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.



The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Whenever the control runs a touch probe cycle, the 3D touch probe approaches the workpiece parallel to the axis. This is also true during an active basic rotation or with a tilted working plane. The machine manufacturer will determine the probing feed rate in a machine parameter.

Further information: "Before you start working with touch probe cycles!", Page 1596

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).

Related topics

- Manual touch probe cycles

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557
- Preset table

  Further information: "Preset table", Page 2035
- Datum table

  Further information: "Datum table", Page 2045
- Reference systems

  Further information: "Reference systems", Page 1010
- Preassigned variables

  Further information: "Preassigned Q parameters", Page 1368

Requirements

- Calibrated workpiece touch probe

  Further information: "Calibrating the workpiece touch probe", Page 1572

  If you are using a HEIDENHAIN touch probe, the software option 17, Touch Probe Functions, is automatically enabled.

Working with an L-shaped stylus

In addition to a **SIMPLE** stylus, probing cycles **444** and **14xx** also support the **L-TYPE** stylus, which is L-shaped. The L-shaped stylus must be calibrated prior to use.

HEIDENHAIN recommends calibrating the stylus with the following cycles:

- Radius calibration: Cycle 460 CALIBRATION OF TS ON A SPHERE (option 17)
- Length calibration: Cycle 461 TS CALIBRATION OF TOOL LENGTH

Stylus orientation must be permitted via **TRACK ON** in the touch probe table. During the probing process, the control orients the L-shaped stylus to the given probing direction. If the probing direction is identical to the tool axis, then the control orients the touch probe to the calibration angle.

- The control does not show the arm of the stylus in the simulation.
- **DCM** (option 40) does not monitor the L-shaped stylus.
- In order to achieve maximum accuracy, the feed rate during calibration must be identical to the feed rate during probing.

Further information: "Touch probe table tchprobe.tp", Page 2022

Notes

The control must be specifically prepared by the machine manufacturer for the use of a 3D touch probe.

While touch probe functions are being executed, the control temporarily disables the **Global Program Settings**.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

Touch probe cycles in the Manual Operation and Electronic Handwheel modes

In the **Setup** application under **Manual** mode, the control provides touch probe cycles that allow you to:

- Set presets
- Probe the angle
- Probe position
- Calibrate the touch probe
- Measure the tool

Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557

Touch probe cycles for automatic operation

Besides the manual touch probe cycles, several cycles are available for a wide variety of applications in automatic operation:

- Automatic measurement of workpiece misalignment
- Automatic determination of the preset
- Automatic workpiece inspection
- Special functions
- Touch probe calibration
- Automatic kinematics measurement
- Automatic tool measurement

Defining touch probe cycles

Like the most recent machining cycles, touch probe cycles with numbers greater than **400** use Q parameters as transfer parameters. Parameters with the same functionality, which the control requires in various cycles, always have the same number: For example, **Q260** is always the clearance height, **Q261** the measuring height, etc.

There are various ways to define the touch probe cycles. Touch probe cycles are programmed in the **Programming** mode of operation.

Inserting via NC function:

Insert
NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- > The control opens a dialog box and prompts you for all required input values.

Inserting via the TOUCH PROBE key:

TOUCH
PROBE

- ▶ Select the **TOUCH PROBE** soft key
- > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- > The control opens a dialog box and prompts you for all required input values.

Navigation in the cycle

Key	Function
▸	Navigation within the cycle: Jump to next parameter
◂	Navigation within the cycle: Jump to previous parameter
▾	Jump to the same parameter in the next cycle
▴	Jump to the same parameter in the previous cycle

For the various cycle parameters, the control provides selectable choices via the action bar or the form.

Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread	
■ Drilling, reaming	Page 488
■ Boring	Page 507
■ Counterboring, centering	
■ Tapping or thread milling	
Pockets/studs/slots	
■ Pocket milling	Page 507
■ Stud milling	
■ Slot milling	
■ Face milling	
Coordinate transformations	
■ Mirroring	Page 1034
■ Rotating	
■ Magnifying / Reducing	
SL cycles	
■ SL (Subcontour List) cycles for the machining of contours that possibly consist of several subcontours	Page 507
■ Cylinder surface machining	Page 1266
■ OCM (Optimized Contour Milling) cycles for combining subcontours to form complex contours	Page 446
Point patterns	
■ Bolt hole circle	Page 431
■ Linear hole pattern	
■ Data Matrix code	
Turning cycles	
■ Area clearance cycles, longitudinal and transverse	Page 744
■ Recess turning cycles, radial and axial	
■ Recessing cycles, radial and axial	
■ Thread cutting cycles	
■ Simultaneous turning cycles	
■ Special cycles	

Cycle group	Further information
Special cycles	
■ Dwell time	Page 1209
■ Program call	Page 507
■ Tolerance	Page 971
■ Oriented spindle stop	Page 1231
■ Engraving	
■ Gear cycles	
■ Interpolation turning	
Grinding cycles	
■ Reciprocating stroke	Page 909
■ Dressing	
■ Compensation cycles	

Measuring cycles

Cycle group	Further information
Rotation	
■ Probing of plane, edge, two circles, beveled edge ■ Basic rotation ■ Two holes or studs ■ Via rotary axis ■ Via C-axis	Page 1600
Preset/Position	
■ Rectangle, inside or outside ■ Circle, inside or outside ■ Corner, inside or outside ■ Center of bolt circle, slot or ridge ■ Touch probe axis or single axis ■ Four holes	Page 1675
Measuring	
■ Angle ■ Circle, inside or outside ■ Rectangle, inside or outside ■ Slot or ridge ■ Bolt hole circle ■ Plane or coordinate	Page 1772
Special cycles	
■ Measuring or measuring in 3D ■ Probing in 3D ■ Fast probing	Page 1831
Calibrating the touch probe	
■ Calibrating the length ■ Calibration in a ring ■ Calibration on a stud ■ Calibration on a sphere	Page 1848
Measuring kinematics	
■ Saving the kinematics ■ Measure kinematics ■ Preset compensation ■ Kinematics grid	Page 1866
Measuring the tool (TT)	
■ Calibrating the TT ■ Tool length, radius or measuring completely ■ Calibrating the IR-TT ■ Lathe tool measurement	Page 1906

31.1.2 Before you start working with touch probe cycles!

General information

In the touch probe table you define the set-up clearance, i.e., how far away from the defined touch point (or the one calculated by the cycle) the control will pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to the one from the touch probe table.

The following can be defined in the touch probe table:

- Type of tool
- Touch probe center offset
- Spindle angle during calibration
- Probing feed rate
- Rapid traverse in probing cycle
- Maximum measuring range
- Set-up clearance
- Feed rate for pre-positioning
- Touch probe orientation
- Serial number
- Reaction in case of collision

Further information: "Touch probe table tchprobe.tp", Page 2022

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.

Positioning logic

Touch probe cycles numbered **400** to **499** or **1400** to **1499** pre-position the touch probe according to the following positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to the clearance height and then positions it in the working plane to the first touch point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the control first positions the touch probe in the working plane to the first touch point, and then in the touch-probe axis directly to the set-up clearance.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- ▶ Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- Reset any coordinate transformations before the cycle call.

- Please note that the units of measure in the measuring log and the return parameters depend on the main program.
- The touch probe cycles **40x** to **43x** will reset an active basic rotation at the beginning of the cycle.
- The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.
- You can apply the misalignment as a workpiece rotation only if a table rotary axis exists on the machine and if its orientation is perpendicular to the workpiece coordinate system **W-CS**.

  Further information: "Comparison of offset and 3D basic rotation", Page 1580

Note regarding machine parameters

- Depending on how the optional machine parameter **chkTiltingAxes** (no. 204600) is set, the control will check during probing whether the position of the rotary axes matches the tilting angles (3D-ROT). If that is not the case, the control displays an error message.

31.1.3 Program defaults for cycles

Entering GLOBAL DEF definitions

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **GLOBAL DEF**
- Select the desired **GLOBAL DEF** function, e.g. **100 GENERAL**
- Enter the required definitions

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:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select and define **GLOBAL DEF**
- Select **Insert NC function** again
- Select the desired cycle, e.g. **200 DRILLING**
- > If the cycle includes global cycle parameters, the control superimposes the selection possibility **PREDEF** in the action bar or in the form as a selection menu.

PREDEF

- Select **PREDEF**
- > The control then 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. Simulate 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

These parameters are valid for all **2xx** machining cycles as well as for Cycles **880, 1017**, **1018**, **1021**, **1022**, **1025** and touch probe cycles **451**, **452**, **453**

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 probing functions

These parameters are valid for all touch probe cycles **4xx** and **14xx** as well as for Cycles **271**, **286**, **287**, **880**, **1021**, **1022**, **1025**, **1271**, **1272**, **1273**, **1278**

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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at 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
```

31.2 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

31.2.1 Overview

The control must be specifically prepared by the machine manufacturer for the use of a 3D touch probe.

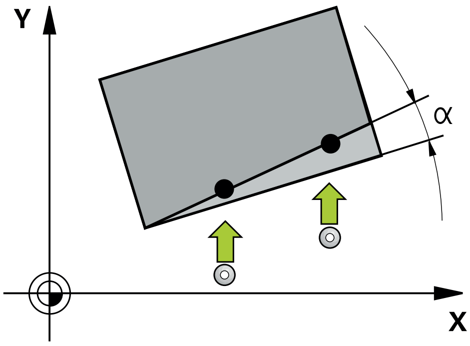
HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

Cycle	Call	Further information
1420 PROBING IN PLANE ■ Automatic measurement using three points ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1612

Cycle		Call	Further information
1410	**PROBING ON EDGE** ■ Automatic measurement using two points ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1618
1411	**PROBING TWO CIRCLES** ■ Automatic measurement using two holes or studs ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1625
1412	**INCLINED EDGE PROBING** ■ Automatic measurement using two points on an inclined edge ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1633
1416	**INTERSECTION PROBING** ■ Automatically determines the intersection with four touch points on two straight lines ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1641
400	**BASIC ROTATION** ■ Automatic measurement using two points ■ Compensation via basic rotation	**DEF**-active	Page 1650
401	**ROT OF 2 HOLES** ■ Automatic measurement using two holes ■ Compensation via basic rotation	**DEF**-active	Page 1653
402	**ROT OF 2 STUDS** ■ Automatic measurement using two studs ■ Compensation via basic rotation	**DEF**-active	Page 1658
403	**ROT IN ROTARY AXIS** ■ Automatic measurement using two points ■ Compensation via rotary table rotation	**DEF**-active	Page 1663
405	**ROT IN C AXIS** ■ Automatic alignment of an angular offset between a hole center and the positive Y axis ■ Compensation via rotary table rotation	**DEF**-active	Page 1668
404	**SET BASIC ROTATION** ■ Setting any basic rotation	**DEF**-active	Page 1673

31.2.2 Fundamentals of touch probe cycles 14xx

Characteristics common to touch probe cycles 14xx for measuring rotations



These cycles can determine rotations. They contain the following:

- Consideration of active machine kinematics
- Semi-automatic probing
- Monitoring of tolerances
- Consideration of 3D calibration
- Simultaneous measurement of rotation and position

Programming and operating notes:

- The probing positions are referenced to the programmed nominal coordinates in the I-CS.
- See the drawing for these nominal positions.
- Before a cycle definition you must program a tool call to define the touch-probe axis.
- The 14xx probing cycles support the **SIMPLE** and **L-TYPE** styli.
- In order to achieve optimal accuracy results with an L-TYPE stylus, HEIDENHAIN recommends that you perform probing and calibration at the same speed. Note the setting of the feed override if it is active during probing.

Explanation of terms

Designation	Short description
Nominal position	Position in the drawing (e.g., position of a hole)
Nominal dimension	Dimension in the drawing (e.g., hole diameter)
Actual position	Measured position (e.g., position of a hole)
Actual dimension	Measured dimension (e.g., hole diameter)
I-CS	I-CS: **Input Coordinate System**
W-CS	W-CS: **Workpiece Coordinate System**
Object	Object to be probed: circle, stud, plane, edge

Evaluation – preset:

- If you want to probe objects in a consistent machining plane or probe objects while TCPM is active, you can program any required shifts as basic transformations in the preset table.
- Rotations can be written to the basic transformations of the preset table as basic rotations or as axial offsets from the first rotary table axis, seen from the workpiece.

> Operating notes:
>
> - When probing, existing 3D calibration data are taken into account. If these calibration data do not exist, deviations might be the result.
> - If you wish to use not only the rotation but also a measured position, then probe as perpendicularly to the surface as possible. The larger the angular error and the larger the ball-tip radius, the larger the positioning error. If the angular errors in the initial angular position are too large, corresponding position errors might be the result.

Logging:

The measured results are recorded in the **TCHPRAUTO.html** file and stored in the Q parameters programmed for this cycle.

The measured deviations are the differences between the measured actual values and the mean tolerance value. If no tolerance has been specified, they refer to the nominal dimension.

The unit of measurement of the main program can be seen in the header of the log.

Semi-automatic mode

If the probing positions relative to the current datum are unknown, you can execute the cycle in semi-automatic mode. In this mode, you can determine the starting position by manually pre-positioning before performing the probing operation.

For this purpose, precede the value for the required nominal position with **"?"**. You can do this by selecting **Name** in the action bar. Depending on the object, you need to define the nominal positions that determine the probing direction, see "Examples."

> Depending on the object, you need to define the nominal positions that determine the probing direction,
>
> Examples:
>
> - Page 1605
> - Page 1606
> - Page 1607

Cycle sequence

Proceed as follows:

- ▶ Run the cycle
- > The control interrupts the NC program.
- > A window opens.
- ▶ Use the axis-direction keys to position the touch probe to the desired touch point

  or
- ▶ Position the touch probe to the desired point using the electronic handwheel
- ▶ Change the probing direction in the window, if necessary

- Select the **NC start** key
- > The control closes the window and performs the first probing operation.
- > If **CLEAR. HEIGHT MODE Q1125** = **1** or **2**, then the control opens a message in the **FN 16** tab, **Status** workspace.
  This message indicates that the mode for traversing to the clearance height is not possible.
- Move the tool to a safe position
- Select the **NC start** key
- > Cycle or program execution is resumed. You may then need to repeat the entire process for further touch points.

NOTICE

Danger of collision!

The control will ignore the programmed values 1 and 2 for Traverse to clearance height when running in semi-automatic mode. Depending on the position of the touch probe, there is danger of collision.

- In semi-automatic mode, manually traverse to a clearance height after every probing operation.

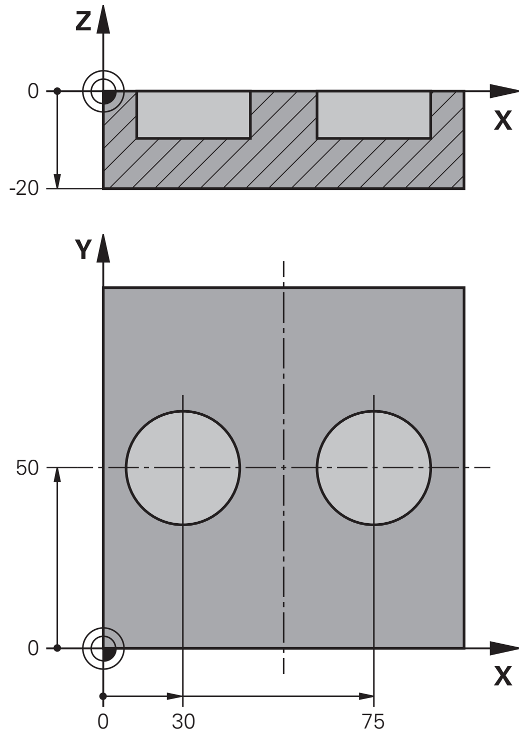
Programming and operating notes:

- See the drawing for these nominal positions.
- Semi-automatic mode is only executed in the machine operating modes, not in the simulation.
- If you did not define a nominal position for a touch point in any direction, the control generates an error message.
- If you did not define a nominal position for a single direction, the control will capture the actual position after probing the object. This means that the measured actual position will subsequently be applied as the nominal position. Consequentially, there is no deviation for this position and thus no position compensation.

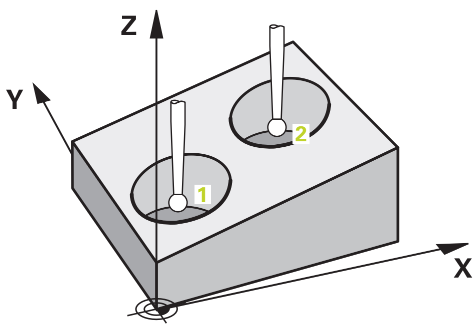
Examples

Important: Specify the **nominal positions** from the drawing!

In the following three examples, the nominal positions from this drawing will be used.



Alignment using two holes



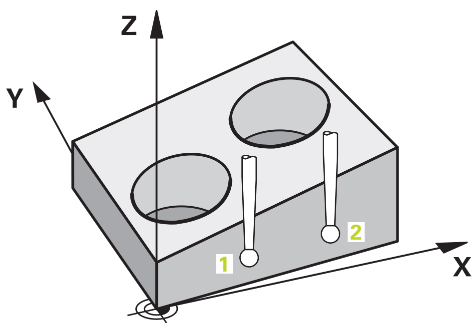
In this example, you will align two holes. Probing is done in the X axis (main axis) and in the Y axis (secondary axis). This means that it is mandatory to define the nominal position from the drawing for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure in this direction.

- **QS1100** = Nominal Position 1 of the main axis is provided, but the workpiece position is not known
- **QS1101** = Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = nominal position 1 in tool axis is unknown
- **QS1103** = Nominal Position 2 of the main axis is provided, but the workpiece position is not known

- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **QS1105** = nominal position 2 in tool axis is unknown

```
11 TCH PROBE 1411 PROBING TWO CIRCLES ~
    QS1100= "?30"          ;1ST POINT REF AXIS ~
    QS1101= "?50"          ;1ST POINT MINOR AXIS ~
    QS1102= "?"            ;1ST POINT TOOL AXIS ~
    Q1116=+10              ;DIAMETER 1 ~
    QS1103= "?75"          ;2ND POINT REF AXIS ~
    QS1104= "?50"          ;2ND POINT MINOR AXIS ~
    QS1105= "?"            ;2ND POINT TOOL AXIS ~
    Q1117=+10              ;DIAMETER 2 ~
    Q1115=+0               ;GEOMETRY TYPE ~
    Q423=+4                ;NO. OF PROBE POINTS ~
    Q325=+0                ;STARTING ANGLE ~
    Q1119=+360             ;ANGULAR LENGTH ~
    Q320=+2                ;SET-UP CLEARANCE ~
    Q260=+100              ;CLEARANCE HEIGHT ~
    Q1125=+2               ;CLEAR. HEIGHT MODE ~
    Q309=+0                ;ERROR REACTION ~
    Q1126=+0               ;ALIGN ROTARY AXIS ~
    Q1120=+0               ;TRANSER POSITION ~
    Q1121=+0               ;CONFIRM ROTATION
```

Alignment through an edge



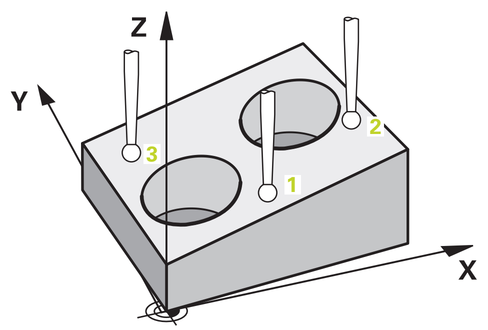
In this example, you will align an edge. Probing is done in the Y axis (secondary axis). This means that it is mandatory to define the nominal position from the drawing for these axes! Nominal positions for the X axis (main axis) and for the Z axis (tool axis) are not required because you will not measure in these directions.

- **QS1100** = nominal position 1 in main axis is unknown
- **QS1101** = Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = nominal position 1 in tool axis is unknown
- **QS1103** = nominal position 2 in main axis is unknown

- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **QS1105** = nominal position 2 in tool axis is unknown

11 TCH PROBE 1410 PROBING ON EDGE ~	
QS1100= "?"	;1ST POINT REF AXIS ~
QS1101= "?0"	;1ST POINT MINOR AXIS ~
QS1102= "?"	;1ST POINT TOOL AXIS ~
QS1103= "?"	;2ND POINT REF AXIS ~
QS1104= "?0"	;2ND POINT MINOR AXIS ~
QS1105= "?"	;2ND POINT TOOL AXIS ~
Q372=+2	;PROBING DIRECTION ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

Alignment via the plane



In this example, you will align a plane. In this case, it is mandatory to define all three nominal positions from the drawing. For angle calculations, it is important that all three axes are taken into account when probing.

- **QS1100** = Nominal Position 1 of the main axis is provided, but the workpiece position is not known
- **QS1101** = Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = Nominal Position 1 of the tool axis is provided, but the workpiece position is not known
- **QS1103** = Nominal Position 2 of the main axis is provided, but the workpiece position is not known
- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **QS1105** = Nominal Position 2 of the tool axis is provided, but the workpiece position is not known
- **QS1106** = Nominal Position 3 of the main axis is provided, but the workpiece position is not known

- **QS1107** = Nominal Position 3 of the secondary axis is provided, but the workpiece position is not known
- **QS1108** = Nominal Position 3 of the tool axis is provided, but the workpiece position is not known

```
11 TCH PROBE 1420 PROBING IN PLANE ~
    QS1100= "?50"        ;1ST POINT REF AXIS ~
    QS1101= "?10"        ;1ST POINT MINOR AXIS ~
    QS1102= "?0"         ;1ST POINT TOOL AXIS ~
    QS1103= "?80"        ;2ND POINT REF AXIS ~
    QS1104= "?50"        ;2ND POINT MINOR AXIS ~
    QS1105= "?0"         ;2ND POINT TOOL AXIS ~
    QS1106= "?20"        ;3RD POINT REF AXIS ~
    QS1107= "?80"        ;3RD POINT MINOR AXIS ~
    QS1108= "?0"         ;3RD POINT TOOL AXIS ~
    Q372=-3              ;PROBING DIRECTION ~
    Q320=+2              ;SET-UP CLEARANCE ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q1125=+2             ;CLEAR. HEIGHT MODE ~
    Q309=+0              ;ERROR REACTION ~
    Q1126=+0             ;ALIGN ROTARY AXIS ~
    Q1120=+0             ;TRANSER POSITION ~
    Q1121=+0             ;CONFIRM ROTATION
```

Evaluation of tolerances

Cycles 14xx also allow you to check tolerance bands. This includes the checking of the position and size of an object.

The following input values with tolerances are possible:

Tolerance	Example
Dimensions	10+0.01-0.015
DIN EN ISO 286-2	10H7
ISO 2768-1	10m

> Pay attention to capitalization when entering tolerances.

If you program a tolerance entry, the control will monitor the tolerance band. The control writes the following statuses to the return parameter **Q183**: Pass, rework, or scrap. If a compensation of the preset is programmed, the control corrects the active preset after probing

The following cycle parameters allow input values with tolerances:

- **Q1100 1ST POINT REF AXIS**
- **Q1101 1ST POINT MINOR AXIS**
- **Q1102 1ST POINT TOOL AXIS**
- **Q1103 2ND POINT REF AXIS**
- **Q1104 2ND POINT MINOR AXIS**
- **Q1105 2ND POINT TOOL AXIS**
- **Q1106 3RD POINT REF AXIS**
- **Q1107 3RD POINT MINOR AXIS**
- **Q1108 3RD POINT TOOL AXIS**
- **Q1116 DIAMETER 1**
- **Q1117 DIAMETER 2**

Program this as follows:

- Start the cycle definition
- Enable the Name selection option in the action bar
- Program nominal position/dimension incl. tolerance
- In the cycle, **QS1116="+8-2-1"** is defined, for example.

> If you program an incorrect tolerance, the control will interrupt machining with an error message.

Cycle sequence

If the actual position is outside the tolerance, the control behaves as follows:

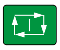
- **Q309 = 0**: The control does not interrupt program run.
- **Q309 = 1**: In the case of scrap or rework, the control interrupts program run with a message.
- **Q309 = 2**: In the case of scrap, the control interrupts program run with a message.

If Q309 = 1 or 2, proceed as follows:

> A window appears. The control displays all of the nominal and actual dimensions of the object.

▶ Press the **CANCEL** button to interrupt the NC program

or



▶ Press **NC start** to resume NC program run

> Please note that the deviations returned by the touch probe cycles are based on the mean tolerance in **Q98x** and **Q99x**. If **Q1120** and **Q1121** are defined, then the values are equivalent to the values used for the compensation. If no automatic evaluation is active, then the control saves the values (based on the mean tolerance) in the intended Q parameter, allowing you to process these values.

Example

- QS1116 = diameter 1, tolerance specified
- QS1117 = diameter 2, tolerance specified

```
11 TCH PROBE 1411PROBING TWO CIRCLES ~
    Q1100=+30            ;1ST POINT REF AXIS ~
    Q1101=+50            ;1ST POINT MINOR AXIS ~
    Q1102=-5             ;1ST POINT TOOL AXIS ~
    QS1116="+8-2-1"      ;DIAMETER 1 ~
    Q1103=+75            ;2ND POINT REF AXIS ~
    Q1104=+50            ;2ND POINT MINOR AXIS ~
    QS1105=-5            ;2ND POINT TOOL AXIS ~
    QS1117="+8-2-1"      ;DIAMETER 2 ~
    Q1115=+0             ;GEOMETRY TYPE ~
    Q423=+4              ;NO. OF PROBE POINTS ~
    Q325=+0              ;STARTING ANGLE ~
    Q1119=+360           ;ANGULAR LENGTH ~
    Q320=+2              ;SET-UP CLEARANCE ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q1125=+2             ;CLEAR. HEIGHT MODE ~
    Q309=2               ;ERROR REACTION ~
    Q1126=+0             ;ALIGN ROTARY AXIS ~
    Q1120=+0             ;TRANSER POSITION ~
    Q1121=+0             ;CONFIRM ROTATION
```

Transferring the actual position

You can determine the actual position in advance and define it as the actual position for the touch probe cycle. Then, both the nominal position and the actual position will be transferred to the object. Based on the difference, the cycle calculates the required compensation values and applies tolerance monitoring.

Program this as follows:

- Define the cycle
- Enable the Name selection option in the action bar
- Program the nominal position with tolerance monitoring as needed
- Program **"@"**
- Program actual position
- \> In the cycle, **QS1100="10+0.02@10.0123"** is defined, for example.

Programming and operating notes:

- If you program @, no probing will be carried out. The control only accounts for the actual and nominal positions.
- You must define the actual position for all three axes: main axis, secondary axis, and tool axis. If you define only one axis with its actual position, an error message will be generated.
- Actual positions can also be defined with Q **Q1900-Q1999**

Example

This feature allows you to do the following:

- Determine a circular pattern based on multiple different objects
- Align a gear based on its center and the position of a tooth

The nominal positions are defined here with tolerance monitoring and actual position.

```
5 TCH PROBE 1410 PROBING ON EDGE ~
    QS1100="10+0.02@10.0123"     ;1ST POINT REF AXIS ~
    QS1101="50@50.0321"          ;1ST POINT MINOR AXIS ~
    QS1102="-10-0.2+0.2@Q1900"   ;1ST POINT TOOL AXIS ~
    QS1103="30+0.02@30.0134"     ;2ND POINT REF AXIS ~
    QS1104="50@50.534"           ;2ND POINT MINOR AXIS ~
    QS1105="-10-0.02@Q1901"      ;2ND POINT TOOL AXIS ~
    Q372=+2                      ;PROBING DIRECTION ~
    Q320=+0                      ;SET-UP CLEARANCE ~
    Q260=+100                    ;CLEARANCE HEIGHT ~
    Q1125=+2                     ;CLEAR. HEIGHT MODE ~
    Q309=+0                      ;ERROR REACTION ~
    Q1126=+0                     ;ALIGN ROTARY AXIS ~
    Q1120=+0                     ;TRANSER POSITION ~
    Q1121=+0                     ;CONFIRM ROTATION
```

31.2.3 Cycle 1420 PROBING IN PLANE

ISO programming
G1420

Application

Touch probe cycle **1420** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

The cycle also offers the following possibilities:

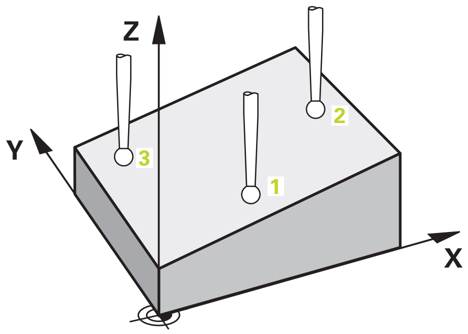
- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1603
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

  Further information: "Evaluation of tolerances", Page 1609
- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1611

Cycle sequence



1 The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programmed touch point **1**.

  Further information: "Positioning logic", Page 1596
2 The control positions the touch probe at **FMAX_PROBE** rapid traverse to the set-up clearance. This clearance is the sum of **Q320**, **SET_UP** and the ball-tip radius. The set-up clearance is considered during probing in every probing direction.
3 The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
5 It then moves in the working plane to touch point **2** to measure the actual value of the second touch point in the plane.

6 The touch probe returns to the clearance height (depending on **Q1125**), then moves in the working plane to touch point **3** and measures the actual position of the third point of the plane.
7 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured position 2 in the main axis, secondary axis, and tool axis
Q956 to **Q958**	Measured position 3 in the main axis, secondary axis, and tool axis
Q961 to **Q963**	Measured spatial angle SPA, SPB, and SPC in the W-CS
Q980 to **Q982**	Measured deviation from the first touch point
Q983 to **Q985**	Measured deviation from the second touch point
Q986 to **Q988**	Third measured deviation of the positions
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Maximum deviation starting from the second touch point
Q972	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Maximum deviation starting from the third touch point

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three touch points are not positioned on a straight line.
- The nominal spatial angle results from the defined nominal positions. The cycle saves the measured spatial angle in parameters **Q961** to **Q963**. For the transfer to the 3D basic rotaton, the control uses the difference between the measured spatial angle and the nominal spatial angle.

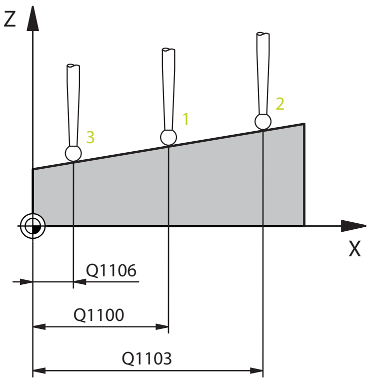
- HEIGHT: HEIDENHAIN recommends avoiding the use of axis angles in this cycle!

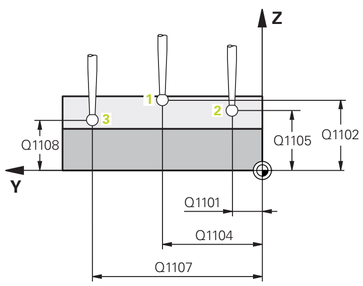
Aligning the rotary table axes:

- Alignment with rotary table axes is possible only if two rotary table axes have been defined in the kinematic model.
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter





Q1100 1st noml. position of ref. axis?

Absolute nominal position of the first touch point in the main axis of the working plane

Input: **-99999.9999...+99999.9999** or **?**, **-**, **+** or **@**

- **?**: Semi-automatic mode, Page 1603
- **-**, **+**: Evaluation of the tolerance, Page 1609
- **@**: Transferring the actual position, Page 1611

Q1101 1st noml. position of minor axis?

Absolute nominal position of the first touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1103 2nd noml. position of ref axis?

Absolute nominal position of the second touch point in the main axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1104 2nd noml. position of minor axis?

Absolute nominal position of the second touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1105 2nd nominal pos. of tool axis?

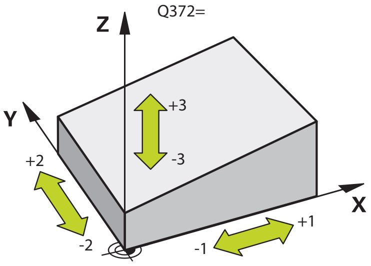
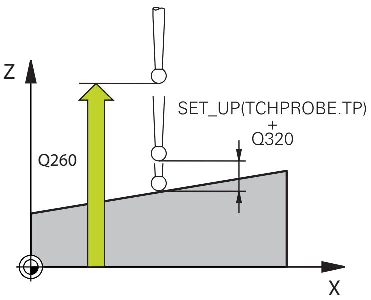
Absolute nominal position of the second touch point in the tool axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1106 3rd noml. position of ref axis?

Absolute nominal position of the third touch point in the main axis of the working plane.

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Help graphic	Parameter
	Q1107 3rd noml. position minor axis? Absolute nominal position of the third touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1108 3rd nominal position tool axis? Absolute nominal position of the third touch point in the tool axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	**Q372 Probe direction (-3 to +3)?** Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	**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** or **PREDEF**
	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** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE** Input: **-1**, **0**, **+1**, **+2**

Help graphic	Parameter
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **2**: Automatically position the rotary axis without orienting the tool tip (**TURN**). Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. **2**: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. **3**: Correction based on 3rd touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 3rd touch point. **4**: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. Input: **0**, **1**, **2**, **3**, **4**
	Q1121 Confirm basic rotation? Define whether the control will use the determined misalignment as a basic rotation: **0**: No basic rotation **1**: Set basic rotation: The control will save the basic rotation Input: **0**, **1**

Example

```
11 TCH PROBE 1420 PROBING IN PLANE ~
    Q1100=+0            ;1ST POINT REF AXIS ~
    Q1101=+0            ;1ST POINT MINOR AXIS ~
    Q1102=+0            ;1ST POINT TOOL AXIS ~
    Q1103=+0            ;2ND POINT REF AXIS ~
    Q1104=+0            ;2ND POINT MINOR AXIS ~
    Q1105=+0            ;2ND POINT TOOL AXIS ~
    Q1106=+0            ;3RD POINT REF AXIS ~
    Q1107=+0            ;3RD POINT MINOR AXIS ~
    Q1108=+0            ;3RD POINT MINOR AXIS ~
    Q372=+1             ;PROBING DIRECTION ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+100           ;CLEARANCE HEIGHT ~
    Q1125=+2            ;CLEAR. HEIGHT MODE ~
    Q309=+0             ;ERROR REACTION ~
    Q1126=+0            ;ALIGN ROTARY AXIS ~
    Q1120=+0            ;TRANSER POSITION ~
    Q1121=+0            ;CONFIRM ROTATION
```

31.2.4 Cycle 1410 PROBING ON EDGE

ISO programming
G1410

Application

Touch probe cycle **1410** allows you to determine workpiece misalignment by probing two points on an edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

The cycle also offers the following possibilities:

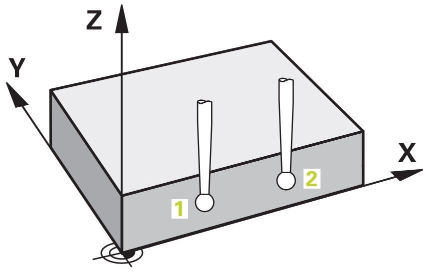
- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1603
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

  Further information: "Evaluation of tolerances", Page 1609
- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1611

Cycle sequence



1 The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programmed touch point **1**.

   Further information: "Positioning logic", Page 1596
2 The control positions the touch probe at **FMAX_PROBE** rapid traverse to the set-up clearance. This clearance is the sum of **Q320**, **SET_UP** and the ball-tip radius. The set-up clearance is considered during probing in every probing direction.
3 The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
4 The control offsets the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.
5 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
6 The touch probe then moves to the next touch point **2** and probes again.
7 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to **Q982**	Measured deviation from the first touch point
Q983 to **Q985**	Measured deviation from the second touch point
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Maximum deviation starting from the second touch point

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

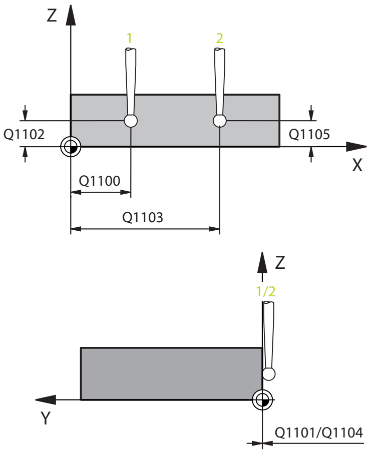
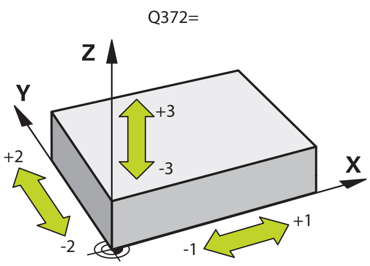
Note about rotary axes:

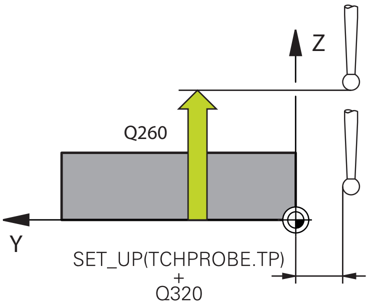
- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (No. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the first touch point in the main axis of the working plane Input: **-99999.9999...+99999.9999** or **?**, **-**, **+** or **@** ■ **?**: Semi-automatic mode, Page 1603 ■ **-**, **+**: Evaluation of the tolerance, Page 1609 ■ **@**: Transferring the actual position, Page 1611
	Q1101 1st noml. position of minor axis? Absolute nominal position of the first touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1103 2nd noml. position of ref axis? Absolute nominal position of the second touch point in the main axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1104 2nd noml. position of minor axis? Absolute nominal position of the second touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	**Q1105 2nd nominal pos. of tool axis?** Absolute nominal position of the second touch point in the tool axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q372 Probe direction (-3 to +3)? Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**

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** or **PREDEF**
	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** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE** Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**

Help graphic	Parameter
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **2**: Automatically position the rotary axis without orienting the tool tip (**TURN**). Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. **2**: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. **3**: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. Input: **0**, **1**, **2**, **3**
	Q1121 CONFIRM ROTATION? Define whether the control should use the determined misalignment: **0**: No basic rotation **1**: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation. **2**: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset. Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 1410 PROBING ON EDGE ~
    Q1100=+0            ;1ST POINT REF AXIS ~
    Q1101=+0            ;1ST POINT MINOR AXIS ~
    Q1102=+0            ;1ST POINT TOOL AXIS ~
    Q1103=+0            ;2ND POINT REF AXIS ~
    Q1104=+0            ;2ND POINT MINOR AXIS ~
    Q1105=+0            ;2ND POINT TOOL AXIS ~
    Q372=+1             ;PROBING DIRECTION ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+100           ;CLEARANCE HEIGHT ~
    Q1125=+2            ;CLEAR. HEIGHT MODE ~
    Q309=+0             ;ERROR REACTION ~
    Q1126=+0            ;ALIGN ROTARY AXIS ~
    Q1120=+0            ;TRANSER POSITION ~
    Q1121=+0            ;CONFIRM ROTATION
```

31.2.5 Cycle 1411 PROBING TWO CIRCLES

ISO programming
G1411

Application

Touch probe cycle **1411** captures the centers of two holes or cylindrical studs and calculates a straight line connecting these centers. The cycle determines the rotation in the working plane based on the difference between the measured angle and the nominal angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

The cycle also offers the following possibilities:

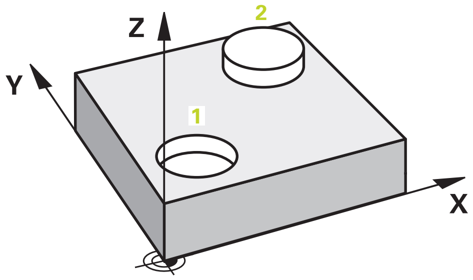
- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1603
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

  Further information: "Evaluation of tolerances", Page 1609
- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1611

Cycle sequence



1 The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programmed center **1**.

   Further information: "Positioning logic", Page 1596
2 The control positions the touch probe at **FMAX_PROBE** rapid traverse to the set-up clearance. This clearance is the sum of **Q320**, **SET_UP** and the ball-tip radius. The set-up clearance is considered during probing in every probing direction.
3 Then the touch probe moves to the entered measuring height **Q1102** at probing feed rate **F** from the touch probe table and probes the first hole or stud center (depending on the number of probes **Q423**).
4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
5 The control positions the touch probe to the entered center or the second hole or the second stud **2**.
6 The controls moves the touch probe to the entered clearance height **Q1105** measures the two holes or stud centers via the probes (based on the number of probes **Q423**).
7 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured circle center point 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured circle center point 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q966 to **Q967**	Measured first and second diameters
Q980 to **Q982**	Measured deviation of the first circle center
Q983 to **Q985**	Measured deviation of the second center
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q996 to **Q997**	Measured deviation of the diameters
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first circle center
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the second circle center
Q973	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from Diameter 1
Q974	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from Diameter 2

Operating note:

- If the hole is too small to achieve the programmed set-up clearance, a window opens. In the window, the control displays the nominal dimension of the hole, the calibrated ball-tip radius, and the achievable set-up clearance.

  You have the following options:

  - If there is no danger of collision, then you can press NC start to run the cycle with the values from the dialog. The active set-up clearance is reduced to the displayed value only for this object.
  - You can cancel the cycle by pressing Cancel.

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

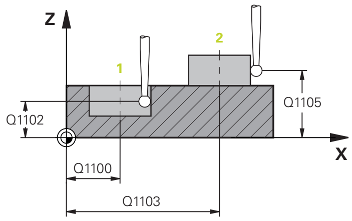
Note about rotary axes:

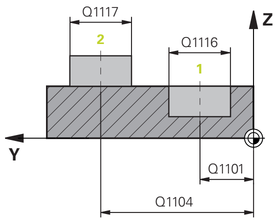
- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (No. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

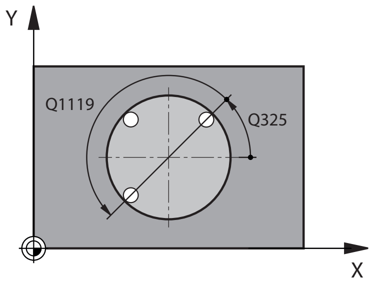
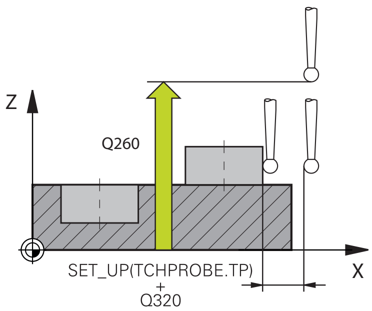
- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters





Help graphic	Parameter
	Q1100 1st noml. position of ref. axis? Absolute nominal position of the first touch point in the main axis of the working plane Input: **-99999.9999...+99999.9999** or **?, -, +** or **@** ■ **?**: Semi-automatic mode, Page 1603 ■ **-, +**: Evaluation of the tolerance, Page 1609 ■ **@**: Transferring the actual position, Page 1611
	Q1101 1st noml. position of minor axis? Absolute nominal position of the first touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1116 Diameter of 1st position? Diameter of the first hole or the first stud Input: **0...9999.9999** or optional input: ■ **"...-...+..."**: Evaluation of the tolerance, Page 1609
	Q1103 2nd noml. position of ref axis? Absolute nominal position of the second touch point in the main axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1104 2nd noml. position of minor axis? Absolute nominal position of the second touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1105 2nd nominal pos. of tool axis? Absolute nominal position of the second touch point in the tool axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Help graphic	Parameter
	Q1117 Diameter of 2nd position? Diameter of the second hole or the second stud Input: **0...9999.9999** or optional input: **"...-...+..."**: Evaluation of the tolerance, Page 1609
	Q1115 Geometry type (0-3)? Type of object to be probed: **0**: Position 1 = hole, and position 2 = hole **1**: Position 1 = stud, and position 2 = stud **2**: Position 1 = hole, and position 2 = stud **3**: Position 1 = stud, and position 2 = hole Input: **0**, **1**, **2**, **3**
	Q423 Number of probes? Number of touch points on the diameter Input: **3**, **4**, **5**, **6**, **7**, **8**
	**Q325 Starting angle?** Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q1119 Arc angular length? Angular range in which the touch points are distributed. Input: **-359.999...+360.000**
	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE** Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **2**: Automatically position the rotary axis without orienting the tool tip (**TURN**). Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. **2**: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. **3**: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	Q1121 CONFIRM ROTATION? Define whether the control should use the determined misalignment: **0**: No basic rotation **1**: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation. **2**: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset. Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 1411 PROBING TWO CIRCLES ~
    Q1100=+0        ;1ST POINT REF AXIS ~
    Q1101=+0        ;1ST POINT MINOR AXIS ~
    Q1102=+0        ;1ST POINT TOOL AXIS ~
    Q1116=+0        ;DIAMETER 1 ~
    Q1103=+0        ;2ND POINT REF AXIS ~
    Q1104=+0        ;2ND POINT MINOR AXIS ~
    Q1105=+0        ;2ND POINT TOOL AXIS ~
    Q1117=+0        ;DIAMETER 2 ~
    Q1115=+0        ;GEOMETRY TYPE ~
    Q423=+4         ;NO. OF PROBE POINTS ~
    Q325=+0         ;STARTING ANGLE ~
    Q1119=+360      ;ANGULAR LENGTH ~
    Q320=+0         ;SET-UP CLEARANCE ~
    Q260=+100       ;CLEARANCE HEIGHT ~
    Q1125=+2        ;CLEAR. HEIGHT MODE ~
    Q309=+0         ;ERROR REACTION ~
    Q1126=+0        ;ALIGN ROTARY AXIS ~
    Q1120=+0        ;TRANSER POSITION ~
    Q1121=+0        ;CONFIRM ROTATION
```

31.2.6 Cycle 1412 INCLINED EDGE PROBING

ISO programming
G1412

Application

Touch probe cycle **1412** allows you to determine workpiece misalignment by probing two points on an inclined edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

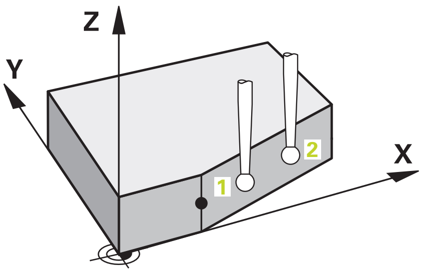
The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1603
- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1611

Cycle sequence



1. The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programmed touch point **1**.

   Further information: "Positioning logic", Page 1596
2. The control positions the touch probe at **FMAX_PROBE** rapid traverse to the set-up clearance. This clearance is the sum of **Q320**, **SET_UP** and the ball-tip radius. The set-up clearance is considered during probing in every probing direction.
3. The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
4. The control retracts the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.
5. If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
6. The touch probe then moves to the touch point **2** and probes again.
7. The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to **Q982**	Measured deviation from the first touch point
Q983 to **Q985**	Measured deviation from the second touch point
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Maximum deviation starting from the second touch point

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program a tolerance in **Q1100**, **Q1101**, or **Q1102**, then this tolerance applies to the programmed nominal positions instead of to the touch points along the inclined edge. Use the parameter **TOLERANCE QS400** to program a tolerance for the surface normal along the inclined edge.

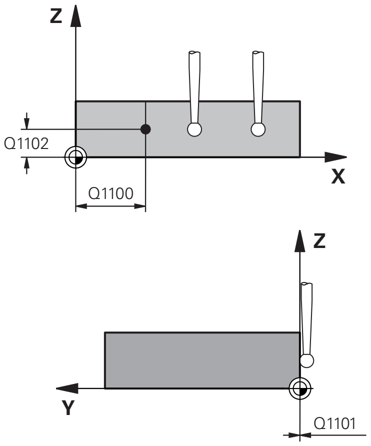
Note about rotary axes:

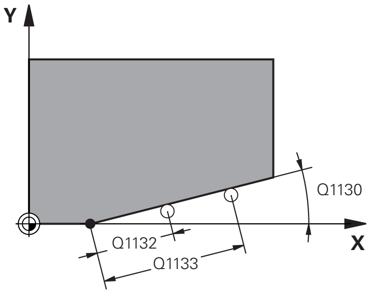
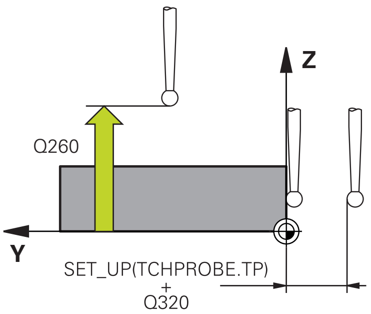
- When you determine the basic rotation in a tilted working plane, keep the following in mind:
  - If the current coordinates of the rotary axes and the defined tilt angles (3D ROT menu) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
  - If the current coordinates of the rotary axes and the defined tilt angles (3D ROT menu) do not match, the working plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** in dependence on the tool axis.
- In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines whether the control checks the matching of the tilting situation. If no check is configured, the control always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position at which the inclined edge begins in the main axis. Input: **-99999.9999...+99999.9999** or **?**, **+**, **-** or **@** ■ **?**: Semi-automatic mode, Page 1603 ■ **-, +**: Evaluation of the tolerance, Page 1609 ■ **@**: Transferring the actual position, Page 1611
	Q1101 1st noml. position of minor axis? Absolute nominal position at which the inclined edge begins in the secondary axis. Input: **-99999.9999...+99999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	QS400 Tolerance value? Tolerance band monitored by the cycle. The tolerance defines the deviation permitted for the surface normals along the inclined edge. The control determines this deviation using the nominal coordinate and the actual coordinate of the workpiece. Examples: ■ **QS400 ="0.4-0.1"**: Upper dimension = Nominal coordinate +0.4; Lower dimension = Nominal coordinate –0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate –0.1" ■ **QS400 =" "**: No monitoring of the tolerance. ■ **QS400 ="0"**: No monitoring of the tolerance. ■ **QS400 ="0.1+0.1"** : No monitoring of the tolerance. Input: Max. **255** characters

Help graphic	Parameter
	**Q1130 Nominal angle for 1st line?** Nominal angle of the first straight line Input: **-180...+180**
	Q1131 Probing direction for 1st line? Probing direction for the first edge: **+1**: Rotates the probing direction by +90° to the nominal angle **Q1130** and probes at right angles to the nominal edge. **-1**: Rotates the probing direction by -90° to the nominal angle **Q1130** and probes at right angles to the nominal edge. Input: **-1**, **+1**
	Q1132 First distance on 1st line? Distance between the beginning of the inclined edge and the first touch point. This value has an incremental effect. Input: **-999.999...+999.999**
	Q1133 Second distance on 1st line? Distance between the beginning of the inclined edge and the second touch point. This value has an incremental effect. Input: **-999.999...+999.999**
	Q1139 Plane for object (1-3)? Plane in which the control interprets the nominal angle **Q1130** and the probing direction **Q1131**. **1**: YZ plane **2**: ZX plane **3**: XY plane Input: **1**, **2**, **3**
	**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** or **PREDEF**
	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** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE** Input: **-1**, **0**, **+1**, **+2**

Help graphic	Parameter
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. **2**: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. **3**: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	Q1121 CONFIRM ROTATION? Define whether the control should use the determined misalignment: **0**: No basic rotation **1**: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation. **2**: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset. Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 1412 INCLINED EDGE PROBING ~
    Q1100=+20            ;1ST POINT REF AXIS ~
    Q1101=+0             ;1ST POINT MINOR AXIS ~
    Q1102=-5             ;1ST POINT TOOL AXIS ~
    QS400="+0.1-0.1"     ;TOLERANCE ~
    Q1130=+30            ;NOMINAL ANGLE, 1ST LINE ~
    Q1131=+1             ;PROBE DIRECTION, 1ST LINE ~
    Q1132=+10            ;FIRST DISTANCE, 1ST LINE ~
    Q1133=+20            ;SECOND DISTANCE, 1ST LINE ~
    Q1139=+3             ;OBJECT PLANE ~
    Q320=+0              ;SET-UP CLEARANCE ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q1125=+2             ;CLEAR. HEIGHT MODE ~
    Q309=+0              ;ERROR REACTION ~
    Q1126=+0             ;ALIGN ROTARY AXIS ~
    Q1120=+0             ;TRANSER POSITION ~
    Q1121=+0             ;CONFIRM ROTATION
```

31.2.7 Cycle 1416 INTERSECTION PROBING

ISO programming
G1416

Application

Touch probe cycle **1416** allows you to determine the intersection of two edges. You can execute the cycle in all three machining planes XY, XZ and YZ. The cycle requires a total of four touch points and two positions per edge. You can select the sequence of the edges as desired.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

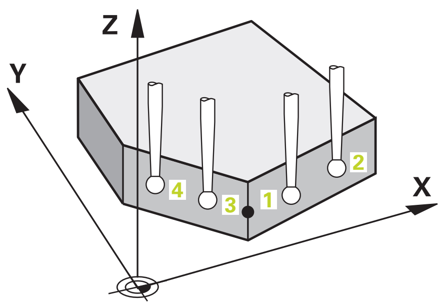
The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1603
- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1611

Cycle sequence



1 The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programmed touch point **1**.

   Further information: "Positioning logic", Page 1596
2 The control positions the touch probe at **FMAX_PROBE** rapid traverse to the set-up clearance. This clearance is the sum of **Q320**, **SET_UP** and the ball-tip radius. The set-up clearance is considered during probing in every probing direction.
3 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
5 The control positions the touch probe to the next touch point.
6 The control positions the touch probe to the entered measuring height **Q1102** and measures the next touch point.
7 The control repeats Steps 4 to 6 until all four touch points are measured.
8 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis and tool axis
Q953 to **Q955**	Measured position 2 in the main axis, secondary axis and tool axis
Q956 to **Q958**	Measured position 3 in the main axis, secondary axis and tool axis
Q959 to **Q960**	Measured intersection in the main axis and secondary axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to **Q982**	Measured deviation of the first touch point in the main axis, auxiliary axis and tool axis
Q983 to **Q985**	Measured deviation of the second touch point in the main axis, auxiliary axis and tool axis
Q986 to **Q988**	Measured deviation of the third touch point in the main axis, auxiliary axis and tool axis
Q989 to **Q990**	Measured deviations of the intersection in the main axis and secondary axis
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have previously programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation from the 1st touch point
Q971	If you have previously programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation from the 2nd touch point
Q972	If you have previously programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation from the 3rd touch point

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

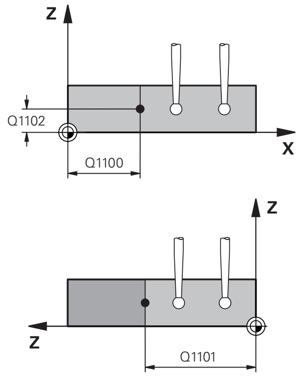
Note about rotary axes:

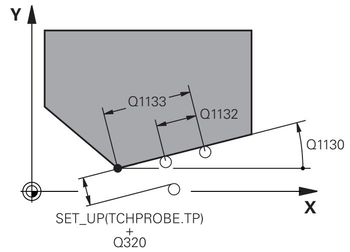
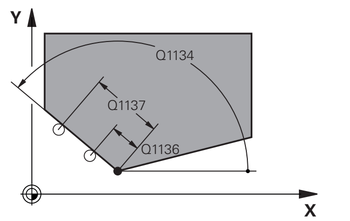
- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (No. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. The basic rotation is then calculated in the **I-CS**.

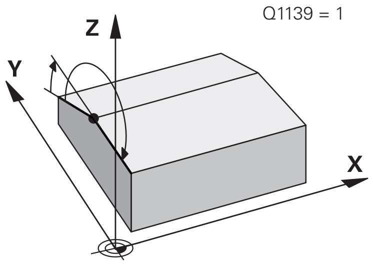
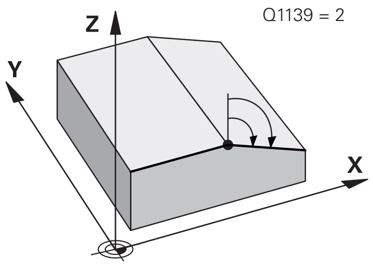
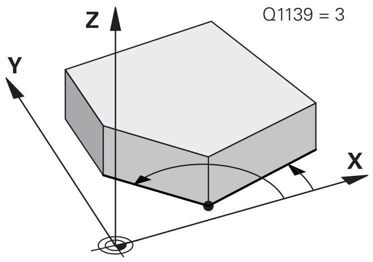
Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position in the main axis at which the two edges intersect. Input: **-99999.9999...+99999.9999** or **?** or **@** ■ **?**: Semi-automatic mode, Page 1603 ■ **@**: Transferring the actual position, Page 1611
	Q1101 1st noml. position of minor axis? Absolute nominal position in the secondary axis at which the two edges intersect. Input: **-99999.9999...+99999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the touch points in the tool axis Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	QS400 Tolerance value? Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normal along the first edge. The control determines the deviation using the nominal coordinates and the actual coordinates of the part. Examples: ■ **QS400 ="0.4-0.1"**: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate −0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate – 0.1" ■ **QS400 =" "**: No monitoring of the tolerance. ■ **QS400 ="0"**: No monitoring of the tolerance. ■ **QS400 ="0.1+0.1"** : No monitoring of the tolerance. Input: Max. **255** characters

Help graphic	Parameter
	**Q1130 Nominal angle for 1st line?** Nominal angle of the first straight line Input: **-180...+180**
	Q1131 Probing direction for 1st line? Probing direction for the first edge: **+1**: Rotates the probing direction by +90° to the nominal angle **Q1130** and probes at right angles to the nominal edge. **-1**: Rotates the probing direction by -90° to the nominal angle **Q1130** and probes at right angles to the nominal edge. Input: **-1**, **+1**
	Q1132 First distance on 1st line? Distance between the intersection and the first touch point on the first edge. This value has an incremental effect. Input: **-999.999...+999.999**
	Q1133 Second distance on 1st line? Distance between the intersection and the second touch point on the first edge. This value has an incremental effect. Input: **-999.999...+999.999**
	QS401 Tolerance value 2? Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normals along the second edge. The control determines this deviation using the nominal coordinate and the actual coordinate of the workpiece. Input: Max. **255** characters
	**Q1134 Nominal angle for 2nd line?** Nominal angle of the first straight line Input: **-180...+180**
	Q1135 Probing direction for 2nd line? Probing direction for the second edge: **+1**: Rotates the probing direction by +90° relative to the nominal angle **Q1134** and probes at right angles relative to the nominal edge. **-1**: Rotates the probing direction by -90° relative to the nominal angle **Q1134**, and probes at right angles relative to the nominal edge. Input: **-1**, **+1**
	Q1136 First distance on 2nd line? Distance between the intersection and the first touch point on the second edge. This value has an incremental effect. Input: **-999.999...+999.999**
	Q1137 Second distance on 2nd line? Distance between the intersection and the second touch point on the second edge. This value has an incremental effect. Input: **-999.999...+999.999**

Help graphic	Parameter
	**Q1139 Plane for object (1-3)?** Plane in which the control interprets the nominal angle **Q1130** and **Q1134**, as well as the probing direction **Q1131** and **Q1135**. **1**: YZ plane **2**: ZX plane **3**: XY plane Input: **1**, **2**, **3**
	**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** or **PREDEF**
	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** or **PREDEF**
	**Q1125 Traverse to clearance height?** Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE** Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**

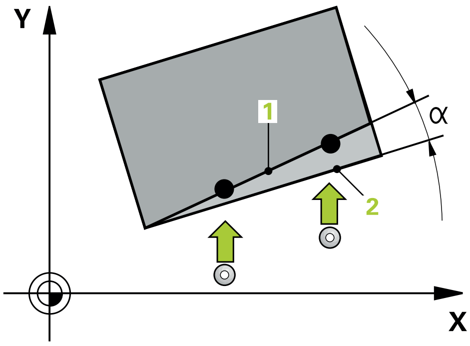
Help graphic	Parameter
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **2**: Automatically position the rotary axis without orienting the tool tip (**TURN**). Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction of the active preset based on the point of intersection. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the intersection. Input: **0**, **1**
	Q1121 CONFIRM ROTATION? Define whether the control should use the determined misalignment: **0**: No basic rotation **1**: Set the basic rotation: The control transfers the misalignment of the first edge to the preset table as a basic transformation. **2**: Execute rotary table rotation: The control transfers the misalignment of the first edge to the preset table as an offset. **3**: Set the basic rotation: The control transfers the misalignment of the second edge to the preset table as a basic transformation. **4**: Execute rotary table rotation: The control transfers the misalignment of the second edge to the preset table as an offset. **5**: Set basic rotation: The control transfers the misalignment from the mean deviations of both edges to the preset table as a basic transformation. **6**: Execute rotary table rotation: The control transfers the misalignment from the mean deviations of both edges to the preset table as an offset. Input: **0**, **1**, **2**, **3**, **4**, **5**, **6**

Example

```
11 TCH PROBE 1416 INTERSECTION PROBING ~
    Q1100=+50            ;1ST POINT REF AXIS ~
    Q1101=+10            ;1ST POINT MINOR AXIS ~
    Q1102=-5             ;1ST POINT TOOL AXIS ~
    QS400="0"            ;TOLERANCE ~
    Q1130=+45            ;NOMINAL ANGLE, 1ST LINE ~
    Q1131=+1             ;PROBE DIRECTION, 1ST LINE ~
    Q1132=+10            ;FIRST DISTANCE, 1ST LINE ~
    Q1133=+25            ;SECOND DISTANCE, 1ST LINE ~
    QS401="0"            ;TOLERANZ 2 ~
    Q1134=+135           ;NOMINAL ANGLE, 2ND LINE ~
    Q1135=-1             ;PROBE DIRECTION, 2ND LINE ~
    Q1136=+10            ;FIRST DISTANCE, 2ND LINE ~
    Q1137=+25            ;SECOND DISTANCE, 2ND LINE ~
    Q1139=+3             ;OBJECT PLANE ~
    Q320=+0              ;SET-UP CLEARANCE ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q1125=+2             ;CLEAR. HEIGHT MODE ~
    Q309=+0              ;ERROR REACTION ~
    Q1126=+0             ;ALIGN ROTARY AXIS ~
    Q1120=+0             ;TRANSER POSITION ~
    Q1121=+0             ;CONFIRM ROTATION
```

31.2.8 Touch probe cycles 4xx: fundamentals

Characteristics common to all touch probe cycles for measuring workpiece misalignment



In Cycles **400**, **401**, and **402**, you can use parameter **Q307 Preset value for rotation angle** to define whether the measurement result will be corrected by a known angle α (see figure). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.

> These cycles do not work with 3D ROT! In such a case, use Cycles **14xx**. **Further information:** "Fundamentals of touch probe cycles 14xx", Page 1602

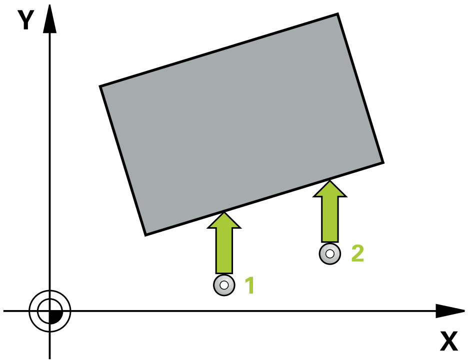
31.2.9 Cycle 400 BASIC ROTATION

ISO programming
G400

Application

Touch probe cycle **400** determines a workpiece misalignment by measuring two points, which must lie on a straight line. With the basic rotation function, the control compensates the measured value.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and performs the basic rotation it determined.

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!

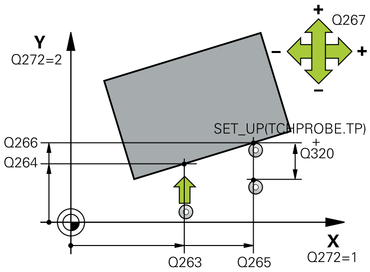
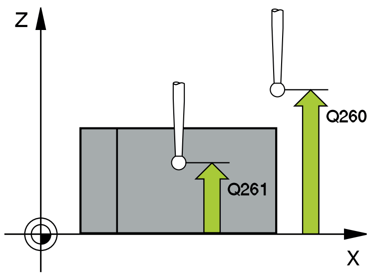
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis Input: **1**, **2**
	Q267 Trav. direction 1 (+1=+ / -1=-)? Direction in which the touch probe will approach the workpiece: **-1**: Negative traverse direction **+1**: Positive traverse direction Input: **-1**, **+1**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q307 Preset value for rotation angle If the misalignment is measured relative to any straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. This value has an absolute effect. Input: **-360.000...+360.000**
	Q305 Preset number in table? Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter **Q305** = 0, the control automatically stores the calculated basic rotation in the ROT menu of the Manual Operation mode. Input: **0...99999**

Example

```
11 TCH PROBE 400 BASIC ROTATION ~
    Q263=+10            ;1ST POINT 1ST AXIS ~
    Q264=+3.5           ;1ST POINT 2ND AXIS ~
    Q265=+25            ;2ND POINT 1ST AXIS ~
    Q266=+2             ;2ND PNT IN 2ND AXIS ~
    Q272=+2             ;MEASURING AXIS ~
    Q267=+1             ;TRAVERSE DIRECTION ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q307=+0             ;PRESET ROTATION ANG. ~
    Q305=+0             ;NUMBER IN TABLE
```

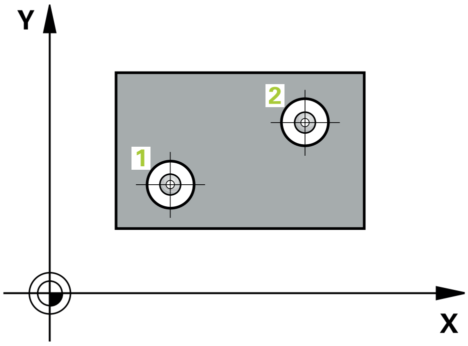
31.2.10 Cycle 401 ROT OF 2 HOLES

ISO programming
G401

Application

Touch probe cycle **401** measures the center points of two holes. The control then calculates the angle between the main axis of the working plane and the line connecting the hole center points. With the basic rotation function, the control compensates for the calculated value. As an alternative, you can also compensate for the determined misalignment by rotating the rotary table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center of the first hole **1**.
   Further information: "Positioning logic", Page 1596
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.

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!

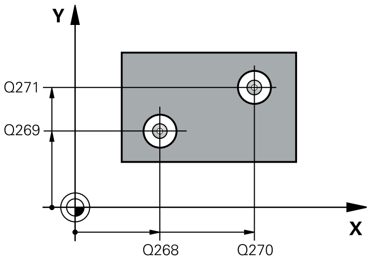
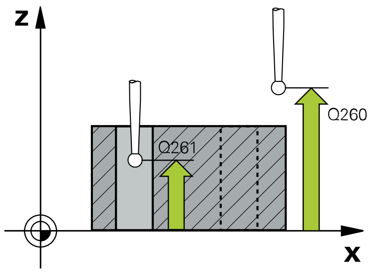
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
  - C for tool axis Z
  - B for tool axis Y
  - A for tool axis X

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q268 1st hole: center in 1st axis?** Center of the first hole in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q269 1st hole: center in 2nd axis? Center of the first hole in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q270 2nd hole: center in 1st axis? Center of the second hole in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q271 2nd hole: center in 2nd axis? Center of the second hole in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+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** or **PREDEF**
	Q307 Preset value for rotation angle If the misalignment is measured relative to any straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. This value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	Q305 Number in table? Enter the number of a row in the preset table. The control will make the corresponding entry in the following row: **Q305** = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0. **Q305** > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**). **Q305 depends on the following parameters:** ■ **Q337** = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, the basic rotation is entered in the **SPC** column). ■ **Q337** = 0 and, at the same time, **Q402** = 1: The parameter **Q305** is not effective. ■ **Q337** = 1: The parameter **Q305** has the effect described above. Input: **0...99999**
	Q402 Basic rotation/alignment (0/1) Define whether the control will set the determined misalignment as a basic rotation or will compensate it by rotating the rotary table: **0**: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**) **1**: Rotate the rotary table: An entry will be made in the corresponding **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column); in addition, the corresponding axis will be rotated Input: **0**, **1**
	Q337 Set to zero after alignment? Define whether the control will set the position display of the corresponding rotary axis to 0 after the alignment: **0**: The position display is not set to 0 after the alignment **1**: After the alignment, the position display is set to 0, provided you have defined **Q402 = 1** Input: **0**, **1**

Example

```
11 TCH PROBE 401 ROT OF 2 HOLES ~
    Q268=-37                ;1ST CENTER 1ST AXIS ~
    Q269=+12                ;1ST CENTER 2ND AXIS ~
    Q270=+75                ;2ND CENTER 1ST AXIS ~
    Q271=+20                ;2ND CENTER 2ND AXIS ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q307=+0                 ;PRESET ROTATION ANG. ~
    Q305=+0                 ;NUMBER IN TABLE ~
    Q402=+0                 ;COMPENSATION ~
    Q337=+0                 ;SET TO ZERO
```

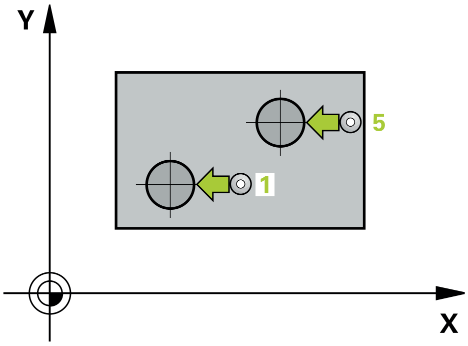
31.2.11 Cycle 402 ROT OF 2 STUDS

ISO programming
G402

Application

Touch probe cycle **402** measures the center points of two cylindrical studs. The control then calculates the angle between the main axis of the working plane and the line connecting the stud center points. With the basic rotation function, the control compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from FMAX column) to touch point **1** of the first stud.
   Further information: "Positioning logic", Page 1596
2 Then the touch probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves along a circular arc between the touch points, each of which is offset by 90°.
3 The touch probe returns to the clearance height and then moves to the touch point **5** of the second stud.
4 The control moves the touch probe to the entered **measuring height 2** and probes four points to determine the center of the second stud.
5 Then the control returns the touch probe to the clearance height and performs the calculated basic rotation.

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!

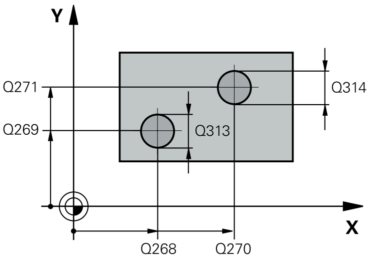
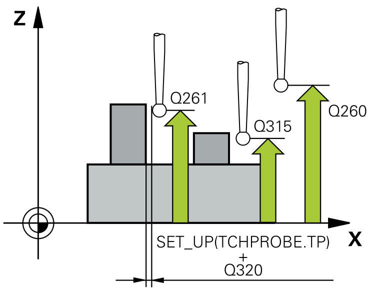
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
  - C for tool axis Z
  - B for tool axis Y
  - A for tool axis X

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q268 1st stud: center in 1st axis?** Center of the first stud in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q269 1st stud: center in 2nd axis? Center of the first stud in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q313 Diameter of stud 1? Approximate diameter of the first stud. Enter a value that is more likely to be too large than too small. Input: **0...99999.9999**
	**Q261 Meas. height stud 1 in TS axis?** Coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 1 will be measured. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q270 2nd stud: center in 1st axis? Center of the second stud in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q271 2nd stud: center in 2nd axis? Center of the second stud in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q314 Diameter of stud 2? Approximate diameter of the second stud. Enter a value that is more likely to be too large than too small. Input: **0...99999.9999**
	Q315 Meas. height stud 2 in TS axis? Coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 2 will be measured. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q307 Preset value for rotation angle If the misalignment is measured relative to any straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. This value has an absolute effect. Input: **-360.000...+360.000**
	Q305 Number in table? Enter the number of a row in the preset table. The control will make the corresponding entry in the following row: **Q305** = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0. **Q305** > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**). **Q305 depends on the following parameters:** ■ **Q337** = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, the basic rotation is entered in the **SPC** column). ■ **Q337** = 0 and, at the same time, **Q402** = 1: The parameter **Q305** is not effective. ■ **Q337** = 1: The parameter **Q305** has the effect described above. Input: **0...99999**

Help graphic	Parameter
	Q402 Basic rotation/alignment (0/1) Define whether the control will set the determined misalignment as a basic rotation or will compensate it by rotating the rotary table: **0**: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**) **1**: Rotate the rotary table: An entry will be made in the corresponding **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column); in addition, the corresponding axis will be rotated Input: **0**, **1**
	Q337 Set to zero after alignment? Define whether the control will set the position display of the corresponding rotary axis to 0 after the alignment: **0**: The position display is not set to 0 after the alignment **1**: After the alignment, the position display is set to 0, provided you have defined **Q402 = 1** Input: **0**, **1**

Example

```
11 TCH PROBE 402 ROT OF 2 STUDS ~
    Q268=-37              ;1ST CENTER 1ST AXIS ~
    Q269=+12              ;1ST CENTER 2ND AXIS ~
    Q313=+60              ;DIAMETER OF STUD 1 ~
    Q261=-5               ;MEAS. HEIGHT STUD 1 ~
    Q270=+75              ;2ND CENTER 1ST AXIS ~
    Q271=+20              ;2ND CENTER 2ND AXIS ~
    Q314=+60              ;DIAMETER OF STUD 2 ~
    Q315=-5               ;MEAS. HEIGHT STUD 2 ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+20              ;CLEARANCE HEIGHT ~
    Q301=+0               ;MOVE TO CLEARANCE ~
    Q307=+0               ;PRESET ROTATION ANG. ~
    Q305=+0               ;NUMBER IN TABLE ~
    Q402=+0               ;COMPENSATION ~
    Q337=+0               ;SET TO ZERO
```

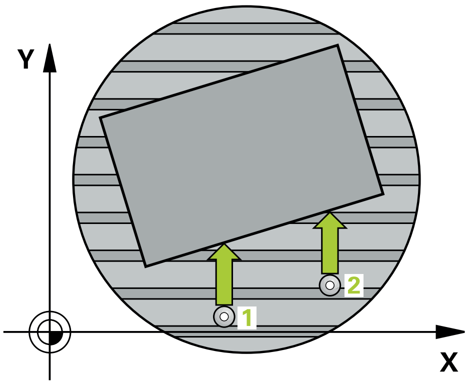
31.2.12 Cycle 403 ROT IN ROTARY AXIS

ISO programming
G403

Application

Touch probe cycle **403** determines a workpiece misalignment by measuring two points, which must lie on a straight line. The control compensates for the determined misalignment by rotating the A, B, or C axis. The workpiece can be clamped in any position on the rotary table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and rotates the rotary axis, which was defined in the cycle, by the measured value. Optionally, you can specify whether the control is to set the determined rotation angle to 0 in the preset table or in the datum table.

Notes

NOTICE
Danger of collision! If the control positions the rotary axis automatically, a collision might occur. ▶ Check for possible collisions between the tool and any elements positioned on the table ▶ Select the clearance height to prevent collisions

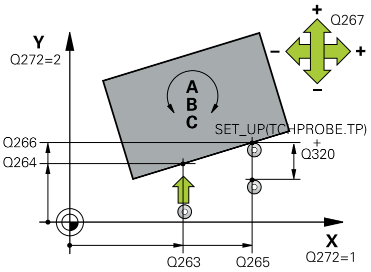
NOTICE
Danger of collision! If you set parameter **Q312** Axis for compensating movement? to 0, then the cycle will automatically determine the rotary axis to be aligned (recommended setting). When doing so, it determines an angle that depends on the sequence of the touch points. The measured angle goes from the first to the second touch point. If you select the A, B or C axis as compensation axis in parameter **Q312**, the cycle determines the angle, regardless of the sequence of the touch points. The calculated angle is in the range of –90° to +90°. There is a risk of collision! ▶ After alignment, check the position of the rotary axis.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**. ▶ Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic | **Parameter**



Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

3: Touch probe axis = measuring axis

Input: **1**, **2**, **3**

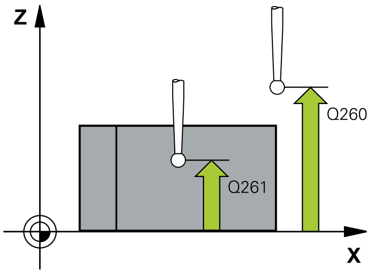
Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

-1: Negative traverse direction

+1: Positive traverse direction

Input: **-1**, **+1**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

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** or **PREDEF**

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** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q312 Axis for compensating movement? Define the rotary axis in which the control will compensate the measured misalignment: **0**: Automatic mode – the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting! **4**: Compensate misalignment with rotary axis A **5**: Compensate misalignment with rotary axis B **6**: Compensate misalignment with rotary axis C Input: **0**, **4**, **5**, **6**
	Q337 Set to zero after alignment? Define whether the control will set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment. **0**: Do not set the angle of the rotary axis to 0 in the table after the alignment **1**: Set the angle of the rotary axis to 0 in the table after the alignment Input: **0**, **1**
	Q305 Number in table? Specify the number of the row in the preset table in which the control will enter the basic rotation. **Q305** = 0: The rotary axis is zeroed in row number 0 of the preset table. The control will make an entry in the **OFFSET** column. In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0. **Q305** > 0: Specify the number of the row in the preset table in which the control will zero the rotary axis. The control will make an entry in the **OFFSET** column of the preset table. **Q305 depends on the following parameters:** ■ **Q337 = 0**: Parameter **Q305** is not effective ■ **Q337 = 1**: Parameter **Q305** has the effect described above ■ **Q312 = 0**: Parameter **Q305** has the effect described above ■ **Q312 > 0**: The entry in **Q305** is ignored. The control will make an entry in the **OFFSET** column, in the row of the preset table that was active when the cycle was called. Input: **0...99999**

Help graphic	Parameter
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **0**: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **0**, **1**
	Q380 Ref. angle in ref. axis? Angle to which the control will align the probed straight line. Only effective if the rotary axis is in automatic mode or if C is selected (**Q312** = 0 or 6). Input: **0...360**

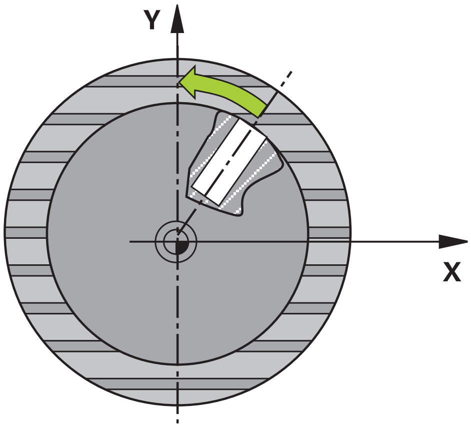
Example

11 TCH PROBE 403 ROT IN ROTARY AXIS ~	
Q263=+0	;1ST POINT 1ST AXIS ~
Q264=+0	;1ST POINT 2ND AXIS ~
Q265=+20	;2ND PNT IN 1ST AXIS ~
Q266=+30	;2ND POINT 2ND AXIS ~
Q272=+1	;MEASURING AXIS ~
Q267=-1	;TRAVERSE DIRECTION ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q312=+0	;COMPENSATION AXIS ~
Q337=+0	;SET TO ZERO ~
Q305=+1	;NUMBER IN TABLE ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q380=+90	;REFERENCE ANGLE

31.2.13 Cycle 405 ROT IN C AXIS

ISO programming
G405

Application

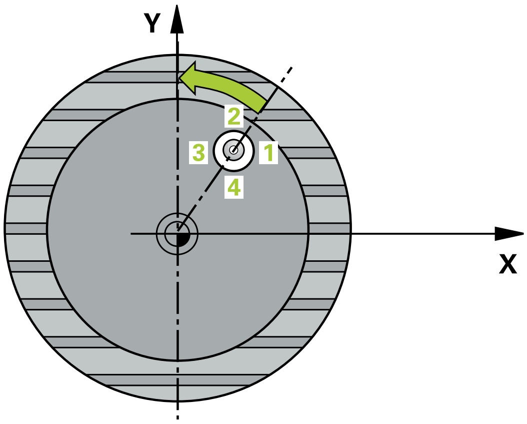


With touch probe cycle **405**, you can measure

- the angular offset between the positive Y axis of the active coordinate system and the center line of a hole
- the angular offset between the nominal position and the actual position of a hole center point

The control compensates for the determined angular offset by rotating the C axis. The workpiece can be clamped in any position on the rotary table, but the Y coordinate of the hole must be positive. If you measure the angular misalignment of the hole with touch probe axis Y (horizontal position of the hole), it may be necessary to execute the cycle more than once because the measuring strategy causes an inaccuracy of approx. 1% of the misalignment.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1596

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves along a circular arc, either at measuring height or at clearance height, to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the calculated hole center.
5 Finally, the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center, after compensation, lies in the direction of the positive Y axis or at the nominal position of the hole center point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in the parameter **Q150**.

Notes

NOTICE
Danger of collision! If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There is a risk of collision! ▶ The pocket/hole must be free of material on the inside ▶ To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**. ▶ Reset any coordinate transformations beforehand.

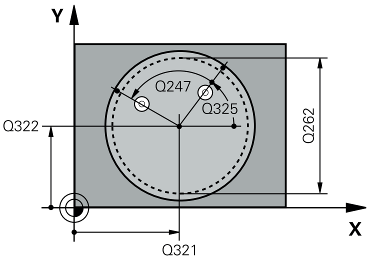
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- The smaller the stepping angle, the less accurately the control can calculate the circle center point. Minimum input value: 5°.

Cycle parameters

Help graphic	Parameter



Q321 Center in 1st axis?

Center of the hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the hole in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point with the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point with the nominal position (angle resulting from the position of the hole center). This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large.

Input: **0...99999.9999**

Q325 Starting angle?

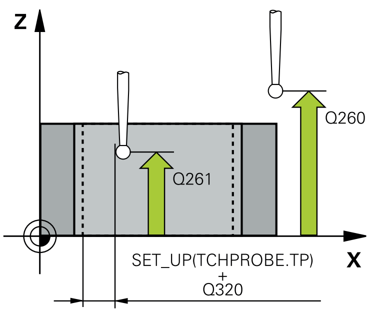
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect.

Input: **-120...+120**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

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** or **PREDEF**

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** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q337 Set to zero after alignment? **0**: Set the display of the C axis to 0 and write to **C_Offset** of the active row of the datum table **> 0**: Write the measured angular offset to the datum table. Row number = value in **Q337**. If a C-axis shift is entered in the datum table, the control adds the measured angular offset with the correct sign, positive or negative. Input: **0...2999**

Example

```
11 TCH PROBE 405 ROT IN C AXIS ~
    Q321=+50        ;CENTER IN 1ST AXIS ~
    Q322=+50        ;CENTER IN 2ND AXIS ~
    Q262=+10        ;NOMINAL DIAMETER ~
    Q325=+0         ;STARTING ANGLE ~
    Q247=+90        ;STEPPING ANGLE ~
    Q261=-5         ;MEASURING HEIGHT ~
    Q320=+0         ;SET-UP CLEARANCE ~
    Q260=+20        ;CLEARANCE HEIGHT ~
    Q301=+0         ;MOVE TO CLEARANCE ~
    Q337=+0         ;SET TO ZERO
```

31.2.14 Cycle 404 SET BASIC ROTATION

ISO programming
G404

Application

With touch probe cycle **404**, you can set any basic rotation automatically during program run or save it to the preset table. You can also use Cycle **404** if you want to reset an active basic rotation.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

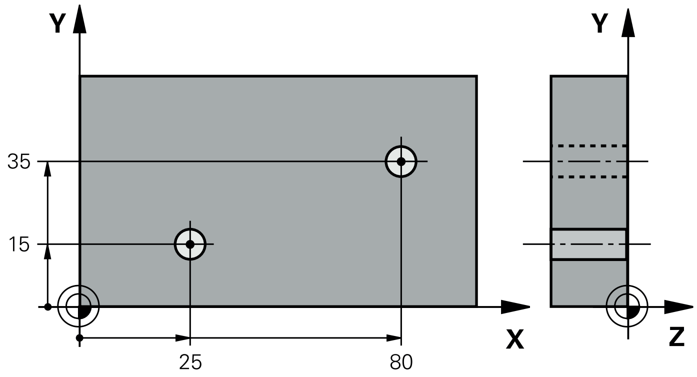
Cycle parameters

Help graphic	Parameter
	Q307 Preset value for rotation angle Angle value at which the basic rotation will be set. Input: **-360.000...+360.000**
	Q305 Preset number in table?: Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter **Q305** = 0 or **Q305** = –1, the control additionally saves the calculated basic rotation in the basic rotation menu (**Probing rot**) of **Manual Operation** mode. **-1**: Overwrite and activate the active preset **0**: Copy the active preset to row 0 of the preset table, write the basic rotation to row 0 of the preset table, and activate preset 0 **> 1**: Save the basic rotation to the specified preset. The preset is not activated. Input: **-1...99999**

Example

```
11 TCH PROBE 404 SET BASIC ROTATION ~
    Q307=+0                  ;PRESET ROTATION ANG. ~
    Q305=-1                  ;NUMBER IN TABLE
```

31.2.15 Example: Determining a basic rotation from two holes



- **Q268** = Center of the 1st hole: X coordinate
- **Q269** = Center of the 1st hole: Y coordinate
- **Q270** = Center of the 2nd hole: X coordinate
- **Q271** = Center of the 2nd hole: Y coordinate
- **Q261** = Coordinate in the touch probe axis in which the measurement is performed
- **Q307** = Angle of the reference line
- **Q402** = Compensation of workpiece misalignment by rotating the table
- **Q337** = Set the display to zero after the alignment

0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 600 Z	
2 TCH PROBE 401 ROT OF 2 HOLES ~	
Q268=+25 ;1ST CENTER 1ST AXIS ~	
Q269=+15 ;1ST CENTER 2ND AXIS ~	
Q270=+80 ;2ND CENTER 1ST AXIS ~	
Q271=+35 ;2ND CENTER 2ND AXIS ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q307=+0 ;PRESET ROTATION ANG. ~	
Q305=+0 ;NUMBER IN TABLE	
Q402=+1 ;COMPENSATION ~	
Q337=+1 ;SET TO ZERO	
3 CALL PGM 35	; Call the part program
4 END PGM TOUCHPROBE MM	

31.3 Touch Probe Cycles: Automatic Preset Measurement

31.3.1 Overview

The control offers cycles for automatic preset measurement.

> 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.

Cycle		Call	Further information
1400	**POSITION PROBING** ■ Measurement of single position ■ Definition of preset, if necessary	**DEF**-active	Page 1677
1401	**CIRCLE PROBING** ■ Measurement of points on the inside or outside of a circle ■ Definition of circle center as preset, if necessary	**DEF**-active	Page 1681
1402	**SPHERE PROBING** ■ Measurement of points on a sphere ■ Definition of sphere center as preset, if necessary	**DEF**-active	Page 1686
1404	**PROBE SLOT/RIDGE** ■ Determine the center of a slot width or ridge width ■ Set the center as a preset if needed	**DEF**-active	Page 1690
1430	**PROBE POSITION OF UNDERCUT** ■ Measure the undercut ■ Measure individual position with L-shaped stylus ■ Set the preset if needed	**DEF**-active	Page 1695
1434	**PROBE SLOT/RIDGE UNDERCUT** ■ Measure the undercut ■ Measure the center of the slot width or ridge width with an L-shaped stylus ■ Set the center as a preset if needed	**DEF**-active	Page 1700
410	**PRESET INSIDE RECTAN** ■ Measurement of inside length and width of a rectangle ■ Definition of rectangle center as preset	**DEF**-active	Page 1707
411	**PRESET OUTS. RECTAN** ■ Measurement of outside length and width of a rectangle ■ Definition of rectangle center as preset	**DEF**-active	Page 1712
412	**PRESET INSIDE CIRCLE**	**DEF**-active	Page 1718

Cycle		Call	Further information
	■ Measurement of any four points on the inside of a circle ■ Definition of circle center as preset		
413	**PRESET OUTS. CIRCLE** ■ Measurement of any four points on the outside of a circle ■ Definition of circle center as preset	**DEF**-active	Page 1724
414	**PRESET OUTS. CORNER** ■ Measurement of two straight lines on the outside ■ Definition of intersection of the lines as preset	**DEF**-active	Page 1730
415	**PRESET INSIDE CORNER** ■ Measurement of two straight lines on the inside ■ Definition of intersection of the lines as preset	**DEF**-active	Page 1736
416	**PRESET CIRCLE CENTER** ■ Measurement of any three holes on a circular hole pattern ■ and defining the circle center as preset	**DEF**-active	Page 1742
417	**PRESET IN TS AXIS** ■ Measurement of any position in the tool axis ■ Definition of any position as preset	**DEF**-active	Page 1748
418	**PRESET FROM 4 HOLES** ■ Measurement of two holes on each line crosswise ■ Definition of the intersection of the lines as preset	**DEF**-active	Page 1752
419	**PRESET IN ONE AXIS** ■ Measurement of any position in a selectable axis ■ Definition of any position in a selectable axis as preset	**DEF**-active	Page 1757
408	**SLOT CENTER PRESET** ■ Measurement of width of an inside slot ■ Definition of slot center as preset	**DEF**-active	Page 1760
409	**RIDGE CENTER PRESET** ■ Measurement of width of an outside ridge ■ Definition of ridge center as preset	**DEF**-active	Page 1765

31.3.2 Fundamentals of touch probe cycles 14xx for setting presets

Characteristics common to all touch probe cycles 14xx for preset setting

Preset and tool axis

The control sets the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Preset setting in
Z	X and Y
Y	Z and X
X	Y and Z

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q9xx**. You can use the parameters in your NC program. Note the table of result parameters listed with every cycle description.

Programming and operating notes:

- The probing positions are based on the programmed nominal coordinates in the I-CS.
- See your drawing for the nominal positions.
- Before defining a cycle, you must program a tool call in order to define the touch-probe axis.
- The 14xx probing cycles support **SIMPLE** and **L-TYPE** styli.
- In order to achieve optimal accuracy results with an L-TYPE stylus, HEIDENHAIN recommends that you perform probing and calibration at the same speed. Note the setting of the feed override if it is active during probing.

31.3.3 Cycle 1400 POSITION PROBING

ISO programming
G1400

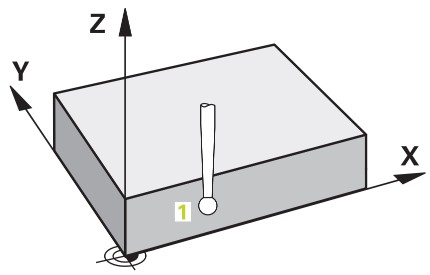
Application

Touch probe cycle **1400** measures any position in a selectable axis. You can apply the result to the active row of the preset table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

Cycle sequence



1 The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programming touch point **1**. During pre-positioning, the control takes the set-up clearance **Q320** into account.

   Further information: "Positioning logic", Page 1596
2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
4 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx for setting presets", Page 1677

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q980 to **Q982**	Measured deviation from the first touch point
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first touch point

Notes

NOTICE

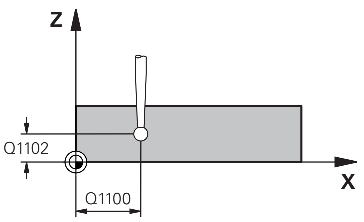
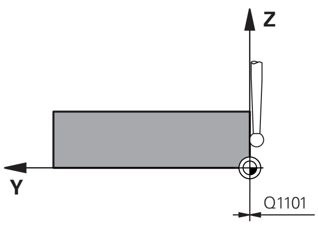
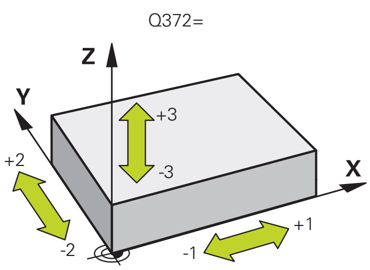
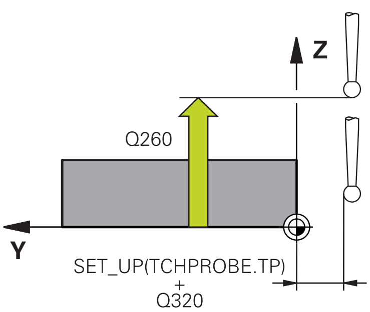
Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the first touch point in the main axis of the working plane Input: **-99999.9999...+99999.9999** or **?**, **-**, **+** or **@** ■ **?**: Semi-automatic mode, Page 1603 ■ **-**, **+**: Evaluation of the tolerance, Page 1609 ■ **@**: Transferring the actual position, Page 1611
	**Q1101 1st noml. position of minor axis?** Absolute nominal position of the first touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	**Q372 Probe direction (-3 to +3)?** Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	**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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0, 1, 2**: Move to the clearance height before and after the touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. Input: **0**, **1**

Example

```
11 TCH PROBE 1400 POSITION PROBING ~
    Q1100=+25                ;1ST POINT REF AXIS ~
    Q1101=+25                ;1ST POINT MINOR AXIS ~
    Q1102=-5                 ;1ST POINT TOOL AXIS ~
    Q372=+0                  ;PROBING DIRECTION ~
    Q320=+0                  ;SET-UP CLEARANCE ~
    Q260=+50                 ;CLEARANCE HEIGHT ~
    Q1125=+1                 ;CLEAR. HEIGHT MODE ~
    Q309=+0                  ;ERROR REACTION ~
    Q1120=+0                 ;TRANSER POSITION
```

31.3.4 Cycle 1401 CIRCLE PROBING

ISO programming
G1401

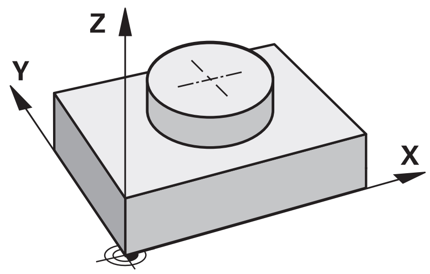
Application

Touch probe cycle **1401** determines the center point of a circular pocket or circular stud. You can transfer the result to the active row of the preset table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

Cycle sequence



1 The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programming touch point **1**. During pre-positioning, the control takes the set-up clearance **Q320** into account.
   Further information: "Positioning logic", Page 1596
2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
4 The control positions the touch probe to the next touch point.
5 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
6 Depending on the definition of **Q423 NO. OF PROBE POINTS**, steps 3 to 5 repeat themselves.
7 The control returns the touch probe to the clearance height **Q260**.
8 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.
   Further information: "Fundamentals of touch probe cycles 14xx for setting presets", Page 1677

Q parameter number	Meaning
Q950 to **Q952**	Measured circle center point in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to **Q982**	Measured deviation of the circle center
Q996	Measured deviation of the diameter
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first circle center
Q973	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from Diameter 1

Notes

NOTICE

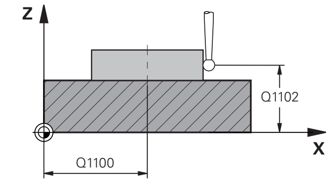
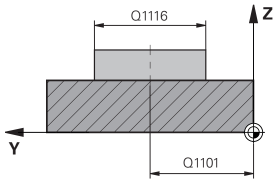
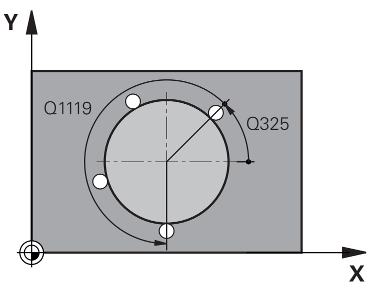
Danger of collision!

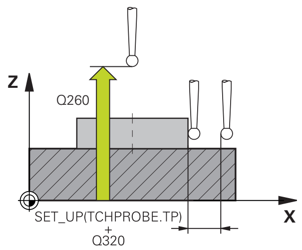
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the center in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**: ■ **"?..."**: Semi-automatic mode, Page 1603 ■ **"...-...+..."**: Evaluation of the tolerance, Page 1609 ■ **"...@..."**: Transferring the actual position, Page 1611
	Q1101 1st noml. position of minor axis? Absolute nominal position of the center in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1116 Diameter of 1st position? Diameter of the first hole or the first stud Input: **0...9999.9999** or optional input: ■ **"...-...+..."**: Evaluation of the tolerance, Page 1609
	Q1115 Geometry type (0/1)? Type of object to be probed: **0**: Hole **1**: Stud Input: **0**, **1**
	Q423 Number of probes? Number of touch points on the diameter Input: **3**, **4**, **5**, **6**, **7**, **8**
	**Q325 Starting angle?** Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q1119 Arc angular length? Angular range in which the touch points are distributed. Input: **-359.999...+360.000**

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** or **PREDEF**
	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** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points **-1**: Do not move to the clearance height. **0**, **1**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. Input: **0**, **1**

Example

11 TCH PROBE 1401 CIRCLE PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS1116=+10	;DIAMETER 1 ~
Q1115=+0	;GEOMETRY TYPE ~
Q423=+3	;NO. OF PROBE POINTS ~
Q325=+0	;STARTING ANGLE ~
Q1119=+360	;ANGULAR LENGTH ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

31.3.5 Cycle 1402 SPHERE PROBING

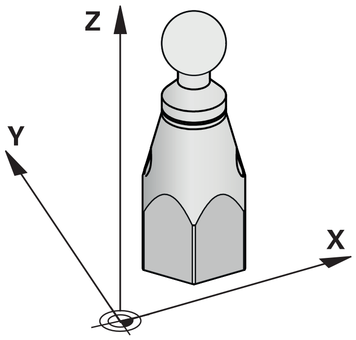
ISO programming

G1402

Application

Touch probe cycle **1402** determines the center point of a sphere. You can apply the result to the active row of the preset table.

Cycle sequence



1. The control positions the touch probe at **FMAX_PROBE** rapid traverse (from the touch probe table) and with positioning logic to the programming touch point **1**. During pre-positioning, the control takes the set-up clearance **Q320** into account.

   Further information: "Positioning logic", Page 1596
2. The control then positions the touch probe to the entered measurement height **Q1102** and performs the first probing procedure at the probing feed rate **F** from the touch probe table.
3. If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
4. The control positions the touch probe to the next touch point.
5. The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
6. Depending on the definition of **Q423** “Number of probe measurements”, steps 3 to 5 repeat themselves.
7. The control moves the touch probe in the tool axis by the set-up clearance to a position above the sphere.
8. The touch probe moves to the center of the sphere and probes another touch point.
9. The touch probe returns to the clearance height **Q260**.
10. The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

    Further information: "Fundamentals of touch probe cycles 14xx for setting presets", Page 1677

Q parameter number	Meaning
Q950 to **Q952**	Measured circle center in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to **Q982**	Measured deviation of the circle center
Q996	Measured deviation of the diameter
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap

Notes

NOTICE

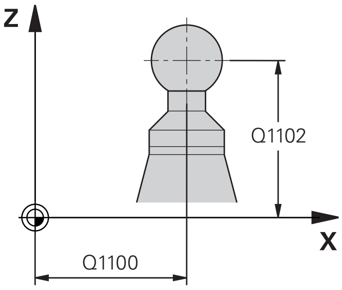
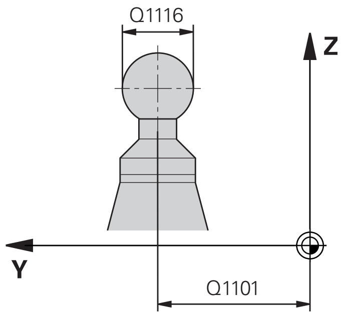
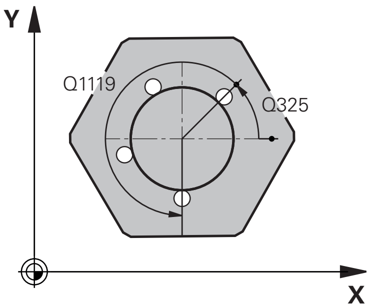
Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you have programmed Cycle **1493 EXTRUSION PROBING** before, the control will ignore it during the execution of Cycle **1402 SPHERE PROBING**.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the center in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**: ■ **"?..."**: Semi-automatic mode, Page 1603 ■ **"...-...+..."**: Evaluation of the tolerance, Page 1609 ■ **"...@..."**: Transferring the actual position, Page 1611
	**Q1101 1st noml. position of minor axis?** Absolute nominal position of the center in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1116 Diameter of 1st position? Diameter of the sphere Input: **0...9999.9999** or optional input (see **Q1100**) ■ **"...-...+..."**: Evaluation of the tolerance, Page 1609
	Q423 Number of probes? Number of touch points on the diameter Input: **3**, **4**, **5**, **6**, **7**, **8**
	**Q325 Starting angle?** Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q1119 Arc angular length? Angular range in which the touch points are distributed. Input: **-359.999...+360.000**
	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** or **PREDEF**

Help graphic	Parameter
	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** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points **-1**: Do not move to the clearance height. **0**, **1**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction of the active preset based on the center of the sphere. The control corrects the active present by the amount of the deviation of the nominal and actual position of the center. Input: **0**, **1**

Example

```
11 TCH PROBE 1402 SPHERE PROBING ~
    Q1100=+25               ;1ST POINT REF AXIS ~
    Q1101=+25               ;1ST POINT MINOR AXIS ~
    Q1102=-5                ;1ST POINT TOOL AXIS ~
    QS1116=+10              ;DIAMETER 1 ~
    Q423=+3                 ;NO. OF PROBE POINTS ~
    Q325=+0                 ;STARTING ANGLE ~
    Q1119=+360              ;ANGULAR LENGTH ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+50                ;CLEARANCE HEIGHT ~
    Q1125=+1                ;CLEAR. HEIGHT MODE ~
    Q309=+0                 ;ERROR REACTION ~
    Q1120=+0                ;TRANSER POSITION
```

31.3.6 Cycle 1404 PROBE SLOT/RIDGE

ISO programming
G1404

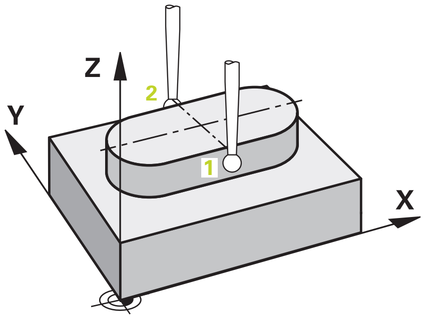
Application

Touch probe cycle **1404** determines the center of the width of a slot or ridge. The control probes the two opposing touch points. The control probes perpendicularly to the angle of rotation of the object to be probed, even if the object to be probed is rotated. You can apply the result to the active row of the preset table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

Cycle sequence



1 At **FMAX_PROBE** rapid traverse from the touch probe table and using positioning logic, the control positions the touch probe to the programmed touch point **1**. During prepositioning, the control takes into account the set-up clearance **Q320**.

   Further information: "Positioning logic", Page 1596

2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.

3 Depending on the selected type of geometry in the parameter **Q1115**, the control proceeds as follows:

   Slot **Q1115=0**:

   - If you program **CLEAR. HEIGHT MODE Q1125** with the value **0**, **1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to **Q260 CLEARANCE HEIGHT**.

   Ridge **Q1115=1**:

   - Independently of **Q1125**, the control positions the touch probe at **FMAX_PROBE** after every touch point back to **Q260 CLEARANCE HEIGHT**.

4 The touch probe moves to the next touch point **2** and performs the second probing procedure at the probing rate **F**.

5 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx for setting presets", Page 1677

Q parameter number	Meaning
Q950 to **Q952**	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to **Q982**	Measured deviation of the center of the slot and ridge
Q998	Measured deviation of the slot width or ridge width
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Measured deviation from the center of the slot or ridge
Q975	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the slot width or ridge width

Notes

NOTICE

Danger of collision!

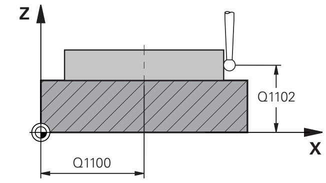
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

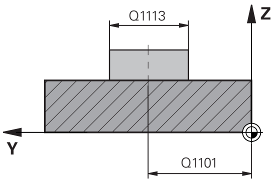
▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter





Q1100 1st noml. position of ref. axis?

Absolute nominal position of the center in the main axis of the working plane.

Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**:

- **"?..."**: Semi-automatic mode, Page 1603
- **"...-...+..."**: Evaluation of the tolerance, Page 1609
- **"...@..."**: Transferring the actual position, Page 1611

Q1101 1st noml. position of minor axis?

Absolute nominal position of the center in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the touch points in the tool axis

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1113 Width of slot/ridge?

Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect.

Input: **0...9999.9999** Or **-** or **+**:

- **"...-...+..."**: Evaluation of the tolerance, Page 1609

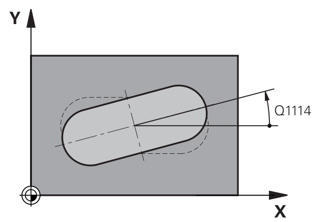
Q1115 Geometry type (0/1)?

Type of object to be probed:

0: Slot

1: Ridge

Input: **0**, **1**



Q1114 Angle of rotation?

Angle about which the slot or the ridge is rotated. The center of rotation is in **Q1100** and **Q1101**. This value has an absolute effect.

Input: **0...359999**

Q320 Set-up clearance?

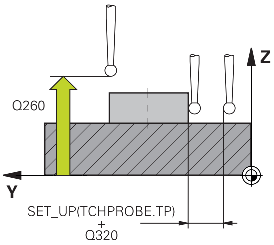
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

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** or **PREDEF**

Help graphic	Parameter
	**Q1125 Traverse to clearance height?** Positioning behavior between the touch points with a slot: **-1**: Do not move to the clearance height. **0, 1**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. The parameter takes effect only with **Q1115=+1** (slot). Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction of the active preset based on the center of the slot or the ridge. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the center. Input: **0**, **1**

Example

```
11 TCH PROBE 1404 PROBE SLOT/RIDGE ~
    Q1100=+25          ;1ST POINT REF AXIS ~
    Q1101=+25          ;1ST POINT MINOR AXIS ~
    Q1102=-5           ;1ST POINT TOOL AXIS ~
    Q1113=+20          ;WIDTH OF SLOT/RIDGE ~
    Q1115=+0           ;GEOMETRY TYPE ~
    Q1114=+0           ;ANGLE OF ROTATION ~
    Q320=+2            ;SET-UP CLEARANCE ~
    Q260=+50           ;CLEARANCE HEIGHT ~
    Q1125=+1           ;CLEAR. HEIGHT MODE ~
    Q309=+0            ;ERROR REACTION ~
    Q1120=+0           ;TRANSER POSITION
```

31.3.7 Cycle 1430 PROBE POSITION OF UNDERCUT

ISO programming
G1430

Application

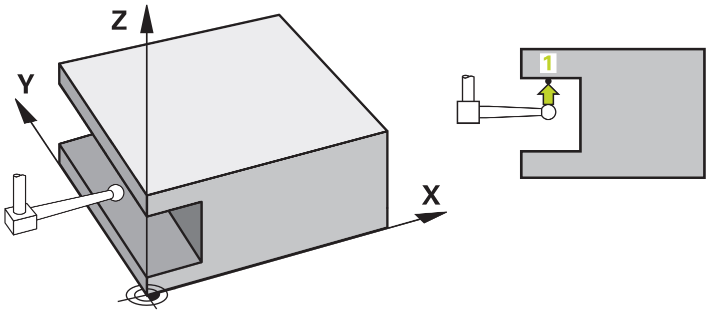
Touch probe cycle **1430** allows a position to be probed with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. You can apply the result of the probing procedure to the active rows of the preset table.

In the main axis and secondary axis, the touch probe is oriented in accordance with the calibration angle. In the tool axis, the touch probe is oriented in accordance with the programmed spindle angle and the calibration angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

Cycle sequence



1 At **FMAX_PROBE** rapid traverse and using positioning logic, the control positions the touch probe to the programmed touch point **1**.

   Pre-position in the machining plane based on the probing direction:
   - **Q372=+/-1**: The pre-position in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position **Q1100**. The radial approach length takes effect in the direction opposite to the probing direction.
   - **Q372=+/-2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the **Q1101**. The radial approach length takes effect in the direction opposite to the probing direction.
   - **Q372=+/-3**: The pre-position of the main axis and secondary axis depends on the direction in which the stylus is oriented. The pre-position is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position. The radial approach length takes effect in the direction opposite to the spindle angle **Q336**.

   Further information: "Positioning logic", Page 1596
2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.
3 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
4 If you program **CLEAR. HEIGHT MODE Q1125** with the value **0**, **1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
5 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx for setting presets", Page 1677

Q parameter number	Meaning
Q950 to **Q952**	Measured position in the main axis, auxiliary axis and tool axis
Q980 to **Q982**	Measured deviation of the position in the main axis, auxiliary axis and tool axis
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the nominal position of the first touch point

Notes

NOTICE

Danger of collision!

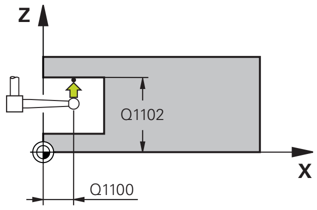
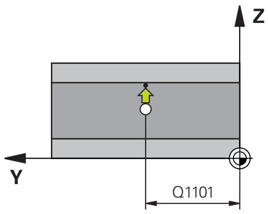
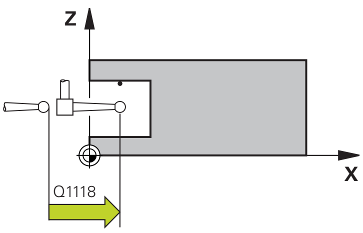
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

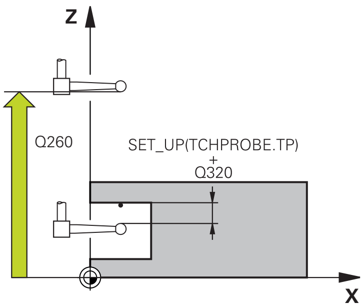
▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle is not intended for L-shaped styli. For simple styli, HEIDENHAIN recommends Cycle **1400 POSITION PROBING**.

  Further information: "Cycle 1400 POSITION PROBING ", Page 1677

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the first touch point in the main axis of the working plane Input: **-99999.9999...+99999.9999** or **?, -, +** or **@** ■ **?**: Semi-automatic mode, Page 1603 ■ **-, +**: Evaluation of the tolerance, Page 1609 ■ **@**: Transferring the actual position, Page 1611
	**Q1101 1st noml. position of minor axis?** Absolute nominal position of the first touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q372 Probe direction (-3 to +3)? Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	Q336 Angle for spindle orientation? Angle at which the control orients the tool prior to the probing procedure. This angle takes effect only during probing in the tool axis (**Q372 = +/- 3**). This value has an absolute effect. Input: **0...360**
	**Q1118 Distance of radial approach?** Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing. If **Q372= +/-1**: Distance is in the direction opposite to the probing direction. If **Q372= +/- 2**: Distance is in the direction opposite to the probing direction. If **Q372= +/-3**: Distance is in the direction opposite to the angle of the spindle **Q336**. This value has an incremental effect. Input: **0...9999.9999**

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** or **PREDEF**
	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** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0, 1, 2**: Move to the clearance height before and after the touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. Input: **0**, **1**

Example

```
11 TCH PROBE 1430 PROBE POSITION OF UNDERCUT ~
    Q1100=+10              ;1ST POINT REF AXIS ~
    Q1101=+25              ;1ST POINT MINOR AXIS ~
    Q1102=-15              ;1ST POINT TOOL AXIS ~
    Q372=+1                ;PROBING DIRECTION ~
    Q336=+0                ;ANGLE OF SPINDLE ~
    Q1118=+20              ;RADIAL APPROACH PATH ~
    Q320=+0                ;SET-UP CLEARANCE ~
    Q260=+50               ;CLEARANCE HEIGHT ~
    Q1125=+1               ;CLEAR. HEIGHT MODE ~
    Q309=+0                ;ERROR REACTION ~
    Q1120=+0               ;TRANSER POSITION
```

31.3.8 Cycle 1434 PROBE SLOT/RIDGE UNDERCUT

ISO programming
G1434

Application

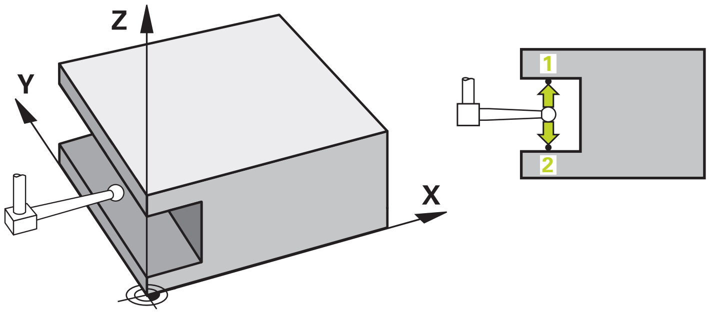
Touch probe cycle **1434** determines the center and width of a slot or a ridge using an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes the two opposing touch points. You can apply the result to the active row of the preset table.

The control orients the touch probe to the calibration angle from the touch probe table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1845

Cycle sequence



1 At **FMAX_PROBE** rapid traverse from the touch probe table, the control positions the touch probe to the pre-position using positioning logic.

   The pre-position in the machining plane depends on the object plane:

   - **Q1139=+1**: The pre-position in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position in **Q1100**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the secondary axis is equivalent to the nominal position.
   - **Q1139=+2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position in **Q1101**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the main axis is equivalent to the nominal position.

   Further information: "Positioning logic", Page 1596

2 The control then positions the touch probe at the entered measuring height **Q1102** and performs the first probing procedure **1** at probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.

3 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.

4 The control positions the touch probe to the next touch point **2** and performs the second probing procedure at probing feed rate **F**.

5 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.

6 If you program the parameter **CLEAR. HEIGHT MODE Q1125** with the value **0** or **1**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

7 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx for setting presets", Page 1677

Q parameter number	Meaning
Q950 to **Q952**	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to **Q982**	Measured deviation of the center of the slot or ridge
Q998	Measured deviation of the slot width or ridge width
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the center of the slot or the ridge
Q975	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the slot width or ridge width

Notes

NOTICE

Danger of collision!

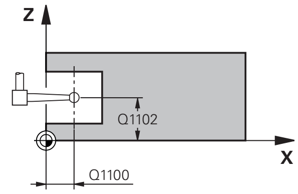
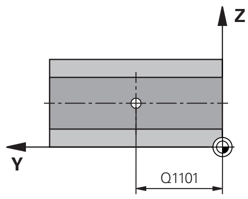
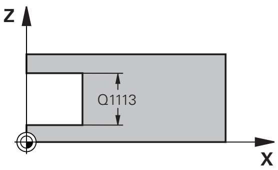
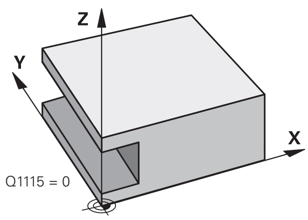
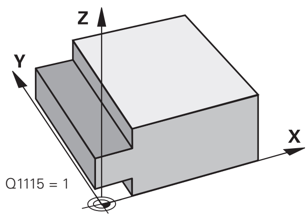
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

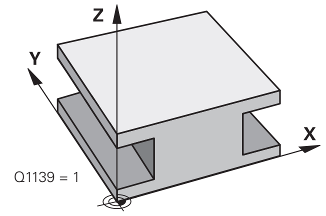
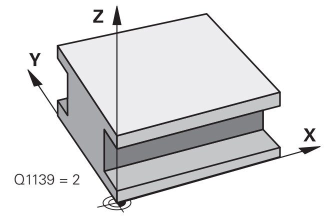
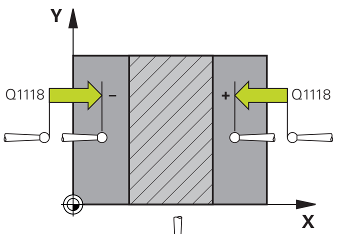
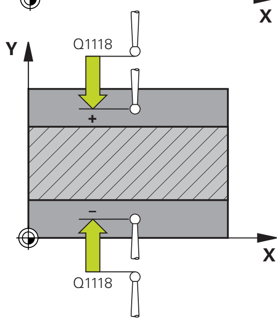
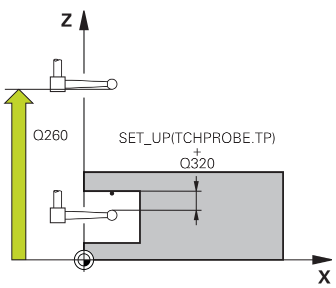
▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program in the radial approach length **Q1118=-0**, then the algebraic sign has no effect. The behavior is identical to +0.
- This cycle is intended for an L-shaped stylus. For simple styli, HEIDENHAIN recommends Cycle **1404 PROBE SLOT/RIDGE**.

  Further information: "Cycle 1404 PROBE SLOT/RIDGE ", Page 1690

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the center in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**: ■ **"?..."**: Semi-automatic mode, Page 1603 ■ **"...-...+..."**: Evaluation of the tolerance, Page 1609 ■ **"...@..."**: Transferring the actual position, Page 1611
	**Q1101 1st noml. position of minor axis?** Absolute nominal position of the center in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	**Q1102 1st nominal position tool axis?** Absolute spindle position of the center in the tool axis Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	**Q1113 Width of slot/ridge?** Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect. Input: **0...9999.9999** Or **-** or **+**: **"...-...+..."**: Evaluation of the tolerance, Page 1609
	**Q1115 Geometry type (0/1)?** Type of object to be probed: **0**: Slot **1**: Ridge Input: **0**, **1**

Help graphic	Parameter
	**Q1139 Object plane (1-2)?** Plane in which the control interprets the probing direction. **1**: YZ plane **2**: ZX plane Input: **1**, **2**
	**Q1118 Distance of radial approach?** Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing. The direction of **Q1118** is equivalent to the probing direction and is in the direction opposite to the algebraic sign. This value has an incremental effect. Input: **-99999.9999...+9999.9999**
	**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** or **PREDEF**
	**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** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior before and after the cycle: **-1**: Do not move to the clearance height. **0, 1**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**
	**Q309 Reaction to tolerance error?** Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction of the active preset based on the center of the slot or the ridge. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the center. Input: **0**, **1**

Example

```
11 TCH PROBE 1434 PROBE SLOT/RIDGE UNDERCUT ~
    Q1100=+25          ;1ST POINT REF AXIS ~
    Q1101=+25          ;1ST POINT MINOR AXIS ~
    Q1102=-5           ;1ST POINT TOOL AXIS ~
    Q1113=+20          ;WIDTH OF SLOT/RIDGE ~
    Q1115=+0           ;GEOMETRY TYPE ~
    Q1139=+1           ;OBJECT PLANE ~
    Q1118=-15          ;RADIAL APPROACH PATH ~
    Q320=+2            ;SET-UP CLEARANCE ~
    Q260=+50           ;CLEARANCE HEIGHT ~
    Q1125=+1           ;CLEAR. HEIGHT MODE ~
    Q309=+0            ;ERROR REACTION ~
    Q1120=+0           ;TRANSER POSITION
```

31.3.9 Fundamentals of touch probe cycles 4xx for preset setting

Characteristics common to all touch probe cycles 4xx for preset setting

> Depending on the setting of the optional **CfgPresetSettings** machine parameter (no. 204600), the control will check during probing whether the position of the rotary axis matches the tilting angles **3-D ROTATION**. If that is not the case, the control displays an error message.

The control offers cycles for automatically determining presets and handling them as follows:

- Setting the calculated values directly as display values
- Writing the calculated values to the preset table
- Writing the calculated values to a datum table

Preset and touch probe axis

The control determines the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Set preset in
Z	X and Y
Y	Z and X
X	Y and Z

Saving the calculated preset

In all cycles for presetting, you can use input parameters **Q303** and **Q305** to define how the control is to save the calculated preset:

- **Q305 = 0, Q303 = 1:**
  The control copies the active preset to row 0, changes it and activates row 0, deleting simple transformations.
- **Q305 not equal to 0, Q303 = 0:**
  The result is written to the datum table, row **Q305**; **activate the datum with TRANS DATUM in the NC program**

  Further information: "Datum shift with TRANS DATUM", Page 1046
- **Q305 not equal to 0, Q303 = 1:**
  The result is written to the preset table, row **Q305**; **use Cycle 247 to activate the preset in the NC program**
- **Q305 not equal to 0, Q303 = -1**

> This combination can only occur if you
> - read in NC programs (containing Cycles **410** to **418**) that were created on a TNC 4xx
> - read in NC programs (containing Cycles **410** to **418**) that were created with an older software version of an iTNC 530
> - did not specifically define the measured-value transfer with parameter **Q303** when defining the cycle
>
> In these cases, the control outputs an error message, since the complete handling of REF-referenced datum tables has changed. You must define a measured-value transfer yourself with parameter **Q303**.

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. You can use these parameters in your NC program. Note the table of result parameters listed with every cycle description.

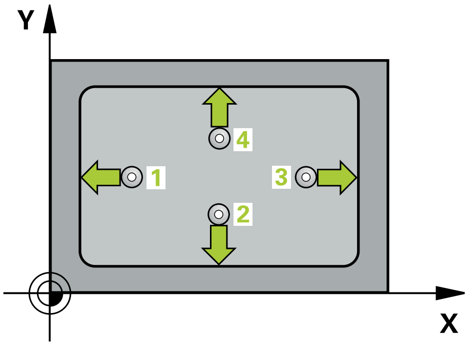
31.3.10 Cycle 410 PRESET INSIDE RECTAN

ISO programming
G410

Application

Touch probe cycle **410** finds the center of a rectangular pocket and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
7 Then the control saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

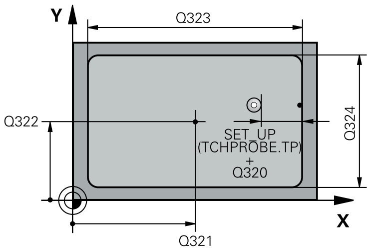
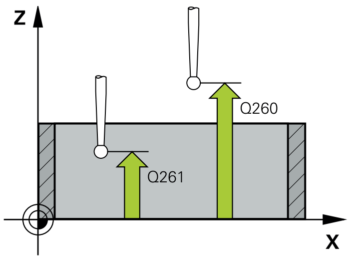
Danger of collision!

If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There is a risk of collision!

- To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides.
- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the pocket in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the pocket in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q323 First side length? Pocket length, parallel to the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q324 Second side length? Pocket length, parallel to the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 CYCL DEF 410 PRESET INSIDE RECTAN ~
    Q321=+50                 ;CENTER IN 1ST AXIS ~
    Q322=+50                 ;CENTER IN 2ND AXIS ~
    Q323=+60                 ;FIRST SIDE LENGTH ~
    Q324=+20                 ;2ND SIDE LENGTH ~
    Q261=-5                  ;MEASURING HEIGHT ~
    Q320=+0                  ;SET-UP CLEARANCE ~
    Q260=+20                 ;CLEARANCE HEIGHT ~
    Q301=+0                  ;MOVE TO CLEARANCE ~
    Q305=+10                 ;NUMBER IN TABLE ~
    Q331=+0                  ;PRESET ~
    Q332=+0                  ;PRESET ~
    Q303=+1                  ;MEAS. VALUE TRANSFER ~
    Q381=+1                  ;PROBE IN TS AXIS ~
    Q382=+85                 ;1ST CO. FOR TS AXIS ~
    Q383=+50                 ;2ND CO. FOR TS AXIS ~
    Q384=+0                  ;3RD CO. FOR TS AXIS ~
    Q333=+1                  ;PRESET
```

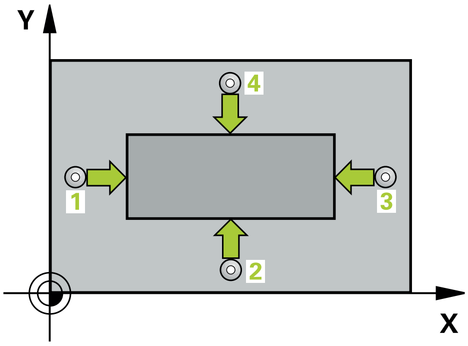
31.3.11 Cycle 411 PRESET OUTS. RECTAN

ISO programming
G411

Application

Touch probe cycle **411** finds the center of a rectangular stud and defines this position as the datum. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1. Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2. Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3. Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4. The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5. The control returns the touch probe to the clearance height.
6. Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
7. Then the control saves the actual values in the Q parameters listed below.
8. If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

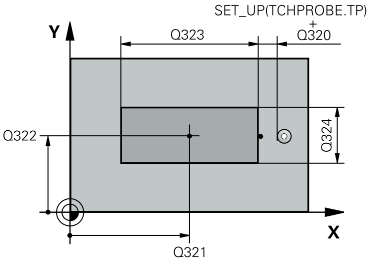
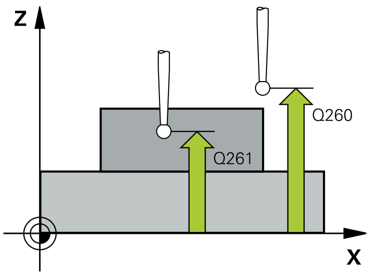
Danger of collision!

To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the stud in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q322 Center in 2nd axis? Center of the stud in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q323 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**
	Q324 Second side length? Length of stud parallel to the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 TCH PROBE 411 PRESET OUTS. RECTAN ~
    Q321=+50                ;CENTER IN 1ST AXIS ~
    Q322=+50                ;CENTER IN 2ND AXIS ~
    Q323=+60                ;FIRST SIDE LENGTH ~
    Q324=+20                ;2ND SIDE LENGTH ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q301=+0                 ;MOVE TO CLEARANCE ~
    Q305=+0                 ;NUMBER IN TABLE ~
    Q331=+0                 ;PRESET ~
    Q332=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER ~
    Q381=+1                 ;PROBE IN TS AXIS ~
    Q382=+85                ;1ST CO. FOR TS AXIS ~
    Q383=+50                ;2ND CO. FOR TS AXIS ~
    Q384=+0                 ;3RD CO. FOR TS AXIS ~
    Q333=+1                 ;PRESET
```

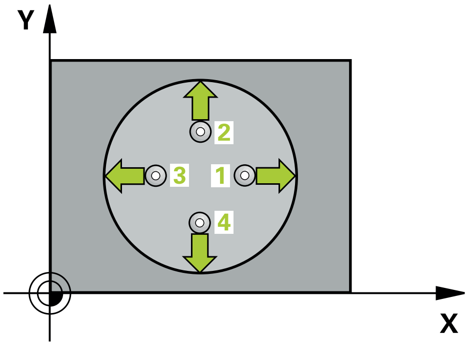
31.3.12 Cycle 412 PRESET INSIDE CIRCLE

ISO programming
G412

Application

Touch probe cycle **412** finds the center of a circular pocket (hole) and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or linearly at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
7 Then the control saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There is a risk of collision!

- The pocket/hole must be free of material on the inside
- To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

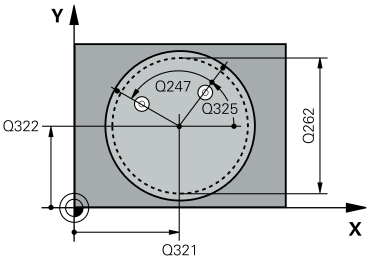
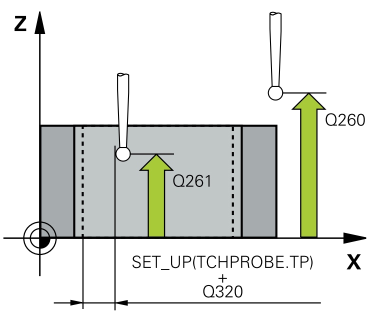
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

Program the stepping angle to be less than 90°

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the pocket in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the pocket in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**

Example

```
11 TCH PROBE 412 PRESET INSIDE CIRCLE ~
    Q321=+50                ;CENTER IN 1ST AXIS ~
    Q322=+50                ;CENTER IN 2ND AXIS ~
    Q262=+75                ;NOMINAL DIAMETER ~
    Q325=+0                 ;STARTING ANGLE ~
    Q247=+60                ;STEPPING ANGLE ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q301=+0                 ;MOVE TO CLEARANCE ~
    Q305=+12                ;NUMBER IN TABLE ~
    Q331=+0                 ;PRESET ~
    Q332=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER ~
    Q381=+1                 ;PROBE IN TS AXIS ~
    Q382=+85                ;1ST CO. FOR TS AXIS ~
    Q383=+50                ;2ND CO. FOR TS AXIS ~
    Q384=+0                 ;3RD CO. FOR TS AXIS ~
    Q333=+1                 ;PRESET ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q365=+1                 ;TYPE OF TRAVERSE
```

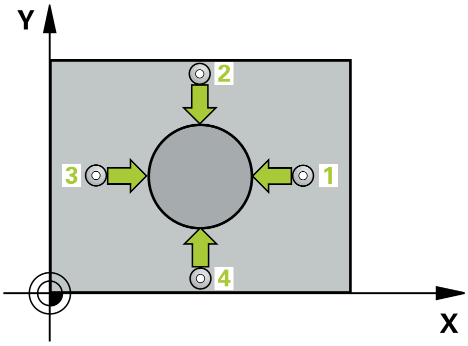
31.3.13 Cycle 413 PRESET OUTS. CIRCLE

ISO programming
G413

Application

Touch probe cycle **413** finds the center of a circular stud and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
7 Then the control saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

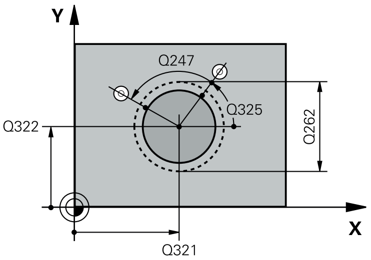
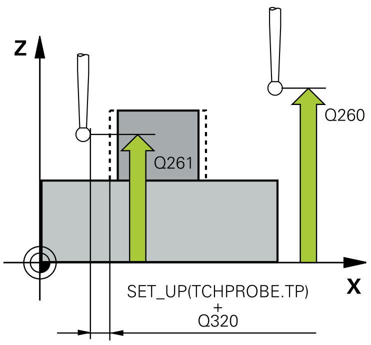
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the nominal diameter of the stud.

- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

- The control will reset an active basic rotation at the beginning of the cycle.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

Program the stepping angle to be less than 90°

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the stud in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q322 Center in 2nd axis? Center of the stud in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**

Example

```
11 TCH PROBE 413 PRESET OUTS. CIRCLE ~
    Q321=+50                 ;CENTER IN 1ST AXIS ~
    Q322=+50                 ;CENTER IN 2ND AXIS ~
    Q262=+75                 ;NOMINAL DIAMETER ~
    Q325=+0                  ;STARTING ANGLE ~
    Q247=+60                 ;STEPPING ANGLE ~
    Q261=-5                  ;MEASURING HEIGHT ~
    Q320=+0                  ;SET-UP CLEARANCE ~
    Q260=+20                 ;CLEARANCE HEIGHT ~
    Q301=+0                  ;MOVE TO CLEARANCE ~
    Q305=+15                 ;NUMBER IN TABLE ~
    Q331=+0                  ;PRESET ~
    Q332=+0                  ;PRESET ~
    Q303=+1                  ;MEAS. VALUE TRANSFER ~
    Q381=+1                  ;PROBE IN TS AXIS ~
    Q382=+85                 ;1ST CO. FOR TS AXIS ~
    Q383=+50                 ;2ND CO. FOR TS AXIS ~
    Q384=+0                  ;3RD CO. FOR TS AXIS ~
    Q333=+1                  ;PRESET ~
    Q423=+4                  ;NO. OF PROBE POINTS ~
    Q365=+1                  ;TYPE OF TRAVERSE
```

31.3.14 Cycle 414 PRESET OUTS. CORNER

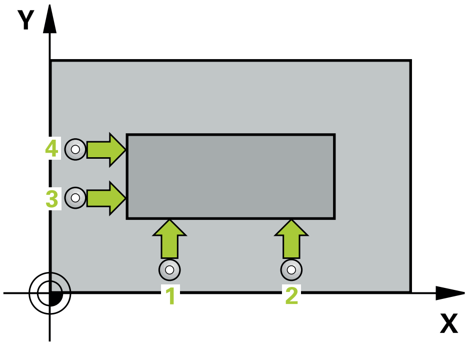
ISO programming

G414

Application

Touch probe cycle **414** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) at touch point **1** (see figure). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.

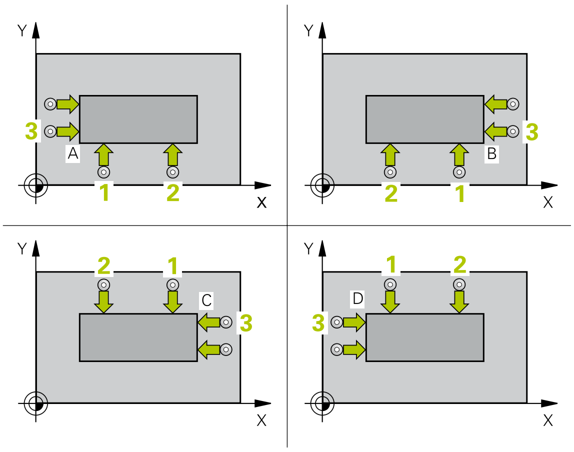
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
7 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

> The control always measures the first line in the direction of the minor axis of the working plane.

Q parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Definition of the corner

By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see the following figure and table below).



Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

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!

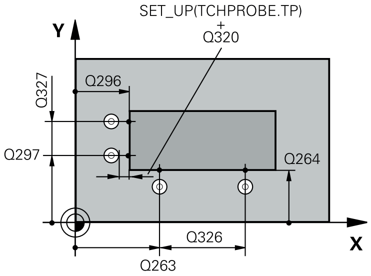
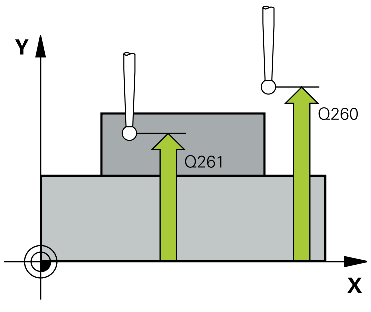
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q326 Spacing in 1st axis? Distance between the first and second measuring points in the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q296 3rd measuring point in 1st axis? Coordinate of the third touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q297 3rd measuring point in 2nd axis? Coordinate of the third touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q327 Spacing in 2nd axis? Distance between third and fourth measuring points in the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**

Help graphic	Parameter
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q304 Execute basic rotation (0/1)? Define whether the control will compensate workpiece misalignment with a basic rotation: **0**: No basic rotation **1**: Basic rotation Input: **0**, **1**
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table: If **Q303 = 1**, the control will write the data to the preset table. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 414 PRESET OUTS. CORNER ~	
Q263=+37	;1ST POINT 1ST AXIS ~
Q264=+7	;1ST POINT 2ND AXIS ~
Q326=+50	;SPACING IN 1ST AXIS ~
Q296=+95	;3RD PNT IN 1ST AXIS ~
Q297=+25	;3RD PNT IN 2ND AXIS ~
Q327=+45	;SPACING IN 2ND AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q304=+0	;BASIC ROTATION ~
Q305=+7	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

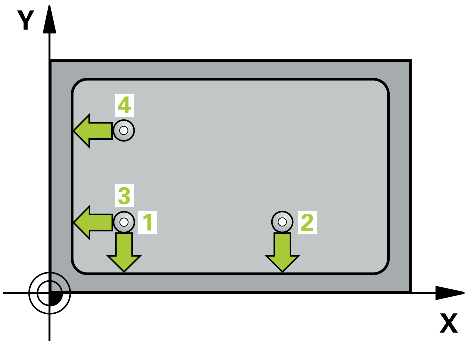
31.3.15 Cycle 415 PRESET INSIDE CORNER

ISO programming
G415

Application

Touch probe cycle **415** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) at touch point **1** (see figure). The control offsets the touch probe in the main axis and the secondary axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius (in the direction opposite the respective traverse direction)

   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The probing direction is derived from the number by which you identify the corner.
3 The touch probe moves to the next touch point **2**; the control offsets the touch probe in the secondary axis by the amount of the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the second probing operation
4 The control positions the touch probe at touch point **3** (same positioning logic as for the first touch point) and performs the probing operation there
5 The touch probe then moves to touch point **4**. The control offsets the touch probe in the main axis by the amount of the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the fourth probing operation
6 The control returns the touch probe to the clearance height.
7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
8 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
9 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

> **i** The control always measures the first line in the direction of the minor axis of the working plane.

Q parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

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!

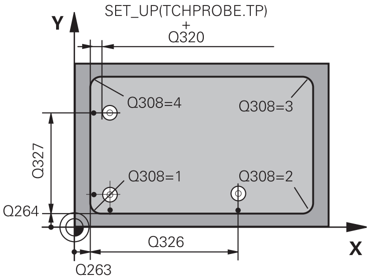
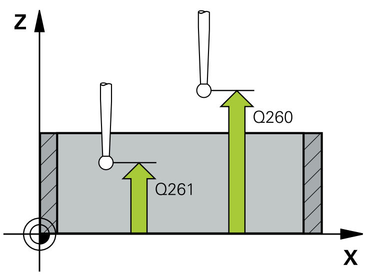
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the corner in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the corner in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q326 Spacing in 1st axis? Distance between the first corner and the second measuring point in the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q327 Spacing in 2nd axis? Distance between the corner and the fourth measuring point in the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q308 Corner? (1/2/3/4) Number identifying the corner at which the control will set the preset. Input: **1**, **2**, **3**, **4**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q304 Execute basic rotation (0/1)? Define whether the control will compensate workpiece misalignment with a basic rotation: **0**: No basic rotation **1**: Basic rotation Input: **0**, **1**
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table: If **Q303 = 1**, the control will write the data to the preset table. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 TCH PROBE 415 PRESET INSIDE CORNER ~
    Q263=+37                ;1ST POINT 1ST AXIS ~
    Q264=+7                 ;1ST POINT 2ND AXIS ~
    Q326=+50                ;SPACING IN 1ST AXIS ~
    Q327=+45                ;SPACING IN 2ND AXIS ~
    Q308=+1                 ;CORNER ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q301=+0                 ;MOVE TO CLEARANCE ~
    Q304=+0                 ;BASIC ROTATION ~
    Q305=+7                 ;NUMBER IN TABLE ~
    Q331=+0                 ;PRESET ~
    Q332=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER ~
    Q381=+1                 ;PROBE IN TS AXIS ~
    Q382=+85                ;1ST CO. FOR TS AXIS ~
    Q383=+50                ;2ND CO. FOR TS AXIS ~
    Q384=+0                 ;3RD CO. FOR TS AXIS ~
    Q333=+1                 ;PRESET
```

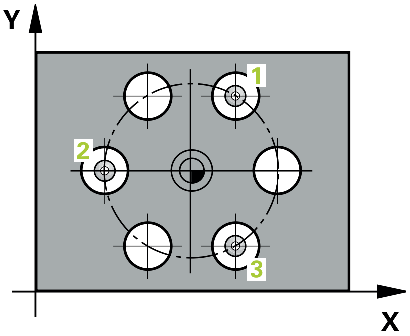
31.3.16 Cycle 416 PRESET CIRCLE CENTER

ISO programming
G416

Application

Touch probe cycle **416** finds the center of a bolt hole circle by measuring three holes, and defines the determined center as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.

   Further information: "Positioning logic", Page 1596
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
7 The control returns the touch probe to the clearance height.
8 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
9 Then the control saves the actual values in the Q parameters listed below.
10 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

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!

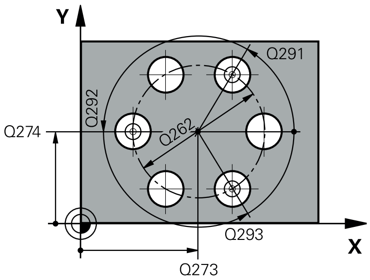
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Bolt hole circle center (nominal value) in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Bolt hole circle center (nominal value) in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input: **0...99999.9999**
	Q291 Polar coord. angle of 1st hole? Polar coordinate angle of the first hole center in the working plane. This value has an absolute effect. Input: **-360.000...+360.000**
	Q292 Polar coord. angle of 2nd hole? Polar coordinate angle of the second hole center in the working plane. This value has an absolute effect. Input: **-360.000...+360.000**
	Q293 Polar coord. angle of 3rd hole? Polar coordinate angle of the third hole center in the working plane. This value has an absolute effect. Input: **-360.000...+360.000**
	Q261 Measuring height in probe axis? Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+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** or **PREDEF**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated bolt-hole center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated bolt-hole circle center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 TCH PROBE 416 PRESET CIRCLE CENTER ~
    Q273=+50                ;CENTER IN 1ST AXIS ~
    Q274=+50                ;CENTER IN 2ND AXIS ~
    Q262=+90                ;NOMINAL DIAMETER ~
    Q291=+34                ;ANGLE OF 1ST HOLE ~
    Q292=+70                ;ANGLE OF 2ND HOLE ~
    Q293=+210               ;ANGLE OF 3RD HOLE ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q305=+12                ;NUMBER IN TABLE ~
    Q331=+0                 ;PRESET ~
    Q332=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER ~
    Q381=+1                 ;PROBE IN TS AXIS ~
    Q382=+85                ;1ST CO. FOR TS AXIS ~
    Q383=+50                ;2ND CO. FOR TS AXIS ~
    Q384=+0                 ;3RD CO. FOR TS AXIS ~
    Q333=+1                 ;PRESET ~
    Q320=+0                 ;SET-UP CLEARANCE
```

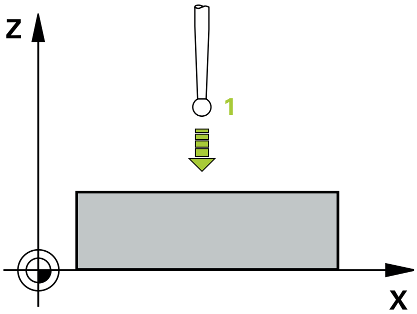
31.3.17 Cycle 417 PRESET IN TS AXIS

ISO programming
G417

Application

Touch probe cycle **417** measures any coordinate in the touch probe axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.

Cycle sequence



1. Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction of the positive touch probe axis.

   Further information: "Positioning logic", Page 1596
2. Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement
3. The control returns the touch probe to the clearance height.
4. Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
5. Then the control saves the actual values in the Q parameters listed below.

Q parameter number	Meaning
Q160	Actual value of measured point

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!

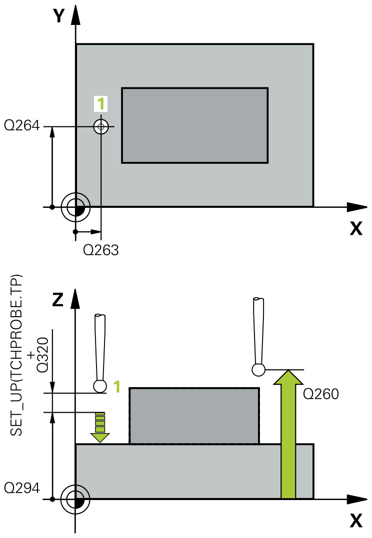
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control sets the preset in this axis.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1706
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Example

```
11 TCH PROBE 417 PRESET IN TS AXIS ~
    Q263=+25                ;1ST POINT 1ST AXIS ~
    Q264=+25                ;1ST POINT 2ND AXIS ~
    Q294=+25                ;1ST POINT 3RD AXIS ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+50                ;CLEARANCE HEIGHT ~
    Q305=+0                 ;NUMBER IN TABLE ~
    Q333=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER
```

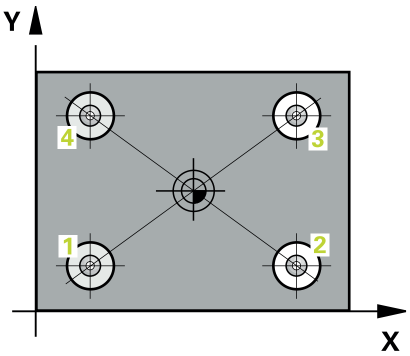
31.3.18 Cycle 418 PRESET FROM 4 HOLES

ISO programming
G418

Application

Touch probe cycle **418** calculates the intersection of the lines connecting two opposite hole center points and sets the preset at the point of intersection. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the center point of the first hole **1**.
   Further information: "Positioning logic", Page 1596
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The control repeats this step for holes **3** and **4**.
6 The control returns the touch probe to the clearance height.
7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
8 The control calculates the preset as the intersection of the lines connecting the centers of holes **1**/**3** and **2**/**4** and saves the actual values in the Q parameters listed below.
9 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

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!

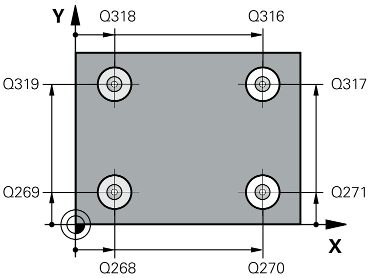
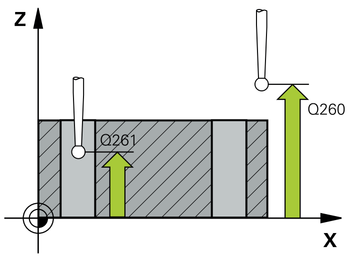
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q268 1st hole: center in 1st axis?** Center of the first hole in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q269 1st hole: center in 2nd axis? Center of the first hole in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q270 2nd hole: center in 1st axis? Center of the second hole in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q271 2nd hole: center in 2nd axis? Center of the second hole in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q316 3rd hole: Center in 1st axis? Center of the third hole in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q317 3rd hole: Center in 2nd axis? Center of the third hole in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q318 4th hole: Center in 1st axis? Center of the fourth hole in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q319 4th hole: Center in 2nd axis? Center of the fourth hole in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+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** or **PREDEF**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row in the preset table or datum table in which the control saves the coordinates of the point of intersection of the connecting lines. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 418 PRESET FROM 4 HOLES ~	
Q268=+20	;1ST CENTER 1ST AXIS ~
Q269=+25	;1ST CENTER 2ND AXIS ~
Q270=+150	;2ND CENTER 1ST AXIS ~
Q271=+25	;2ND CENTER 2ND AXIS ~
Q316=+150	;3RD CENTER 1ST AXIS ~
Q317=+85	;3RD CENTER 2ND AXIS ~
Q318=+22	;4TH CENTER 1ST AXIS ~
Q319=+80	;4TH CENTER 2ND AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q305=+12	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+0	;PRESET

31.3.19 Cycle 419 PRESET IN ONE AXIS

ISO programming
G419

Application

Touch probe cycle **419** measures any coordinate in the a selectable axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.

Cycle sequence

1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the programmed probing direction.
   Further information: "Positioning logic", Page 1596
2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
3 The control returns the touch probe to the clearance height.
4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

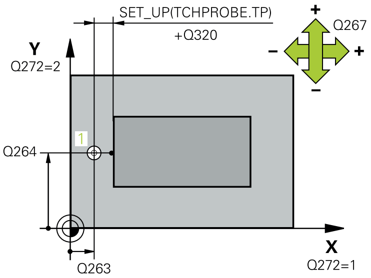
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you want to save the preset in several axes in the preset table, you can use Cycle **419** several times in a row. However, you also have to reactivate the preset number after every run of Cycle **419**. If you work with preset 0 as active preset, this process is not required.
- The control will reset an active basic rotation at the beginning of the cycle.

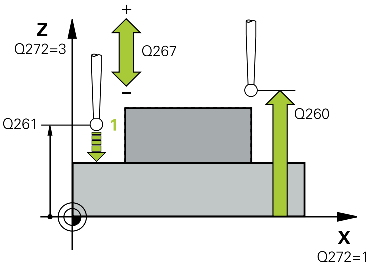
Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic





Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

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** or **PREDEF**

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** or **PREDEF**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

3: Touch probe axis = measuring axis

Axis assignment

Active touch probe axis: Q272 = 3	Correspond-ing main axis: Q272 = 1	Corresponding secondary axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

Input: **1**, **2**, **3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

-1: Negative traverse direction

+1: Positive traverse direction

Input: **-1**, **+1**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1706
	Q333 New preset? Coordinate at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are uploaded; (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1705) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Example

```
11 TCH PROBE 419 PRESET IN ONE AXIS ~
    Q263=+25                 ;1ST POINT 1ST AXIS ~
    Q264=+25                 ;1ST POINT 2ND AXIS ~
    Q261=+25                 ;MEASURING HEIGHT ~
    Q320=+0                  ;SET-UP CLEARANCE ~
    Q260=+50                 ;CLEARANCE HEIGHT ~
    Q272=+1                  ;MEASURING AXIS ~
    Q267=+1                  ;TRAVERSE DIRECTION ~
    Q305=+0                  ;NUMBER IN TABLE ~
    Q333=+0                  ;PRESET ~
    Q303=+1                  ;MEAS. VALUE TRANSFER
```

31.3.20 Cycle 408 SLOT CENTER PRESET

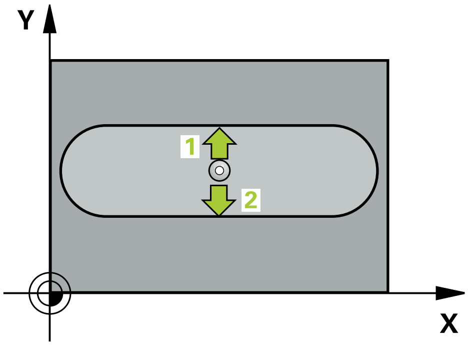
ISO programming

G408

Application

Touch probe cycle **408** finds the center of a slot and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height.
5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
6 Then the control saves the actual values in the Q parameters listed below.
7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

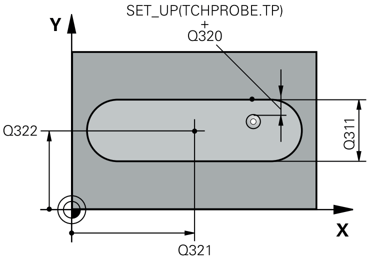
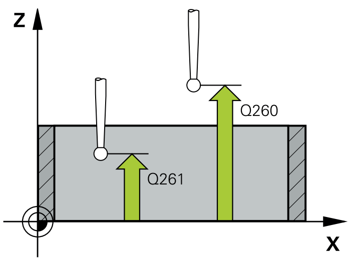
Danger of collision!

If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch probe does not return to the clearance height between the two measuring points. There is a risk of collision!

- To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width.
- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the slot in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the slot in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q311 Width of slot? Width of the slot, regardless of its position in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis Input: **1**, **2**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q405 New preset? Coordinate in the measuring axis at which the control will set the calculated slot center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **0**: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **0**, **1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 TCH PROBE 408 SLOT CENTER PRESET ~
    Q321=+50            ;CENTER IN 1ST AXIS ~
    Q322=+50            ;CENTER IN 2ND AXIS ~
    Q311=+25            ;SLOT WIDTH ~
    Q272=+1             ;MEASURING AXIS ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q305=+10            ;NUMBER IN TABLE ~
    Q405=+0             ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+85            ;1ST CO. FOR TS AXIS ~
    Q383=+50            ;2ND CO. FOR TS AXIS ~
    Q384=+0             ;3RD CO. FOR TS AXIS ~
    Q333=+1             ;PRESET
```

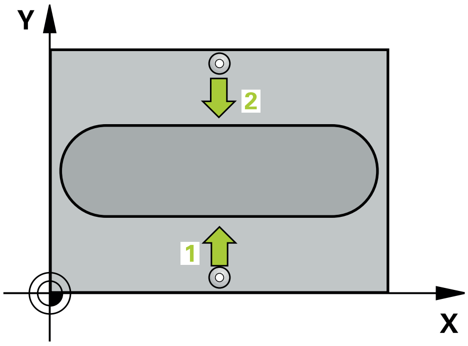
31.3.21 Cycle 409 RIDGE CENTER PRESET

ISO programming
G409

Application

Touch probe cycle **409** finds the center of a ridge and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.
4 The control returns the touch probe to the clearance height.
5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1705)
6 Then the control saves the actual values in the Q parameters listed below.
7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

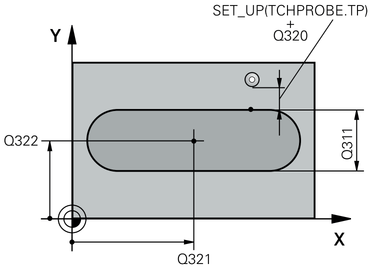
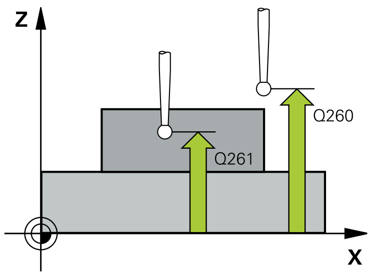
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the ridge in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the ridge in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q311 Ridge width? Width of the ridge, regardless of its position in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis Input: **1**, **2**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**

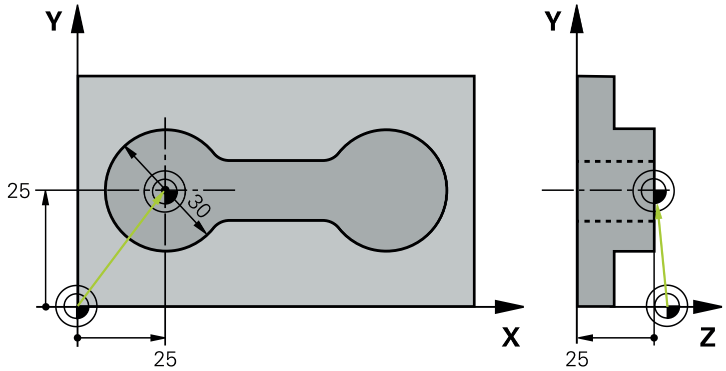
Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. **Further information:** "Saving the calculated preset", Page 1706 Input: **0...99999**
	Q405 New preset? Coordinate in the measuring axis at which the control will set the calculated ridge center. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **0**: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **0**, **1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 TCH PROBE 409 RIDGE CENTER PRESET ~
    Q321=+50            ;CENTER IN 1ST AXIS ~
    Q322=+50            ;CENTER IN 2ND AXIS ~
    Q311=+25            ;RIDGE WIDTH ~
    Q272=+1             ;MEASURING AXIS ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q305=+10            ;NUMBER IN TABLE ~
    Q405=+0             ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+85            ;1ST CO. FOR TS AXIS ~
    Q383=+50            ;2ND CO. FOR TS AXIS ~
    Q384=+0             ;3RD CO. FOR TS AXIS ~
    Q333=+1             ;PRESET
```

31.3.22 Example: Presetting at center of a circular segment and on top surface of workpiece

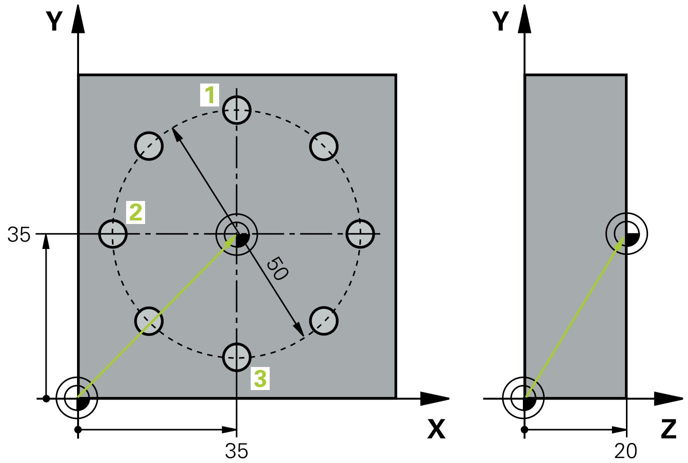


- **Q325** = Polar coordinate angle for touch point 1
- **Q247** = Stepping angle for calculating the touch points 2 to 4
- **Q305** = Write to row number 5 of the preset table
- **Q303** = Write the calculated preset to the preset table
- **Q381** = Also set the preset in the touch probe axis
- **Q365** = Move on circular path between measuring points

```
0  BEGIN PGM 413 MM
1  TOOL CALL "TOUCH_PROBE" Z
2  TCH PROBE 413 PRESET OUTS. CIRCLE ~
    Q321=+25               ;CENTER IN 1ST AXIS ~
    Q322=+25               ;CENTER IN 2ND AXIS ~
    Q262=+30               ;NOMINAL DIAMETER ~
    Q325=+90               ;STARTING ANGLE ~
    Q247=+45               ;STEPPING ANGLE ~
    Q261=-5                ;MEASURING HEIGHT ~
    Q320=+2                ;SET-UP CLEARANCE ~
    Q260=+50               ;CLEARANCE HEIGHT ~
    Q301=+0                ;MOVE TO CLEARANCE ~
    Q305=+5                ;NUMBER IN TABLE ~
    Q331=+0                ;PRESET ~
    Q332=+10               ;PRESET ~
    Q303=+1                ;MEAS. VALUE TRANSFER ~
    Q381=+1                ;PROBE IN TS AXIS ~
    Q382=+25               ;1ST CO. FOR TS AXIS ~
    Q383=+25               ;2ND CO. FOR TS AXIS ~
    Q384=+0                ;3RD CO. FOR TS AXIS ~
    Q333=+0                ;PRESET ~
    Q423=+4                ;NO. OF PROBE POINTS ~
    Q365=+0                ;TYPE OF TRAVERSE
3  END PGM 413 MM
```

31.3.23 Example: Presetting on top surface of workpiece and at center of a bolt hole circle

The control will write the measured bolt-hole circle center to the preset table so that it may be used at a later time.



- **Q291** = Polar coordinate angle for first hole center **1**
- **Q292** = Polar coordinate angle for second hole center **2**
- **Q293** = Polar coordinate angle for third hole center **3**
- **Q305** = Write center of bolt hole circle (X and Y) to row 1
- **Q303** = In the preset table **PRESET.PR**, save the calculated preset referenced to the machine-based coordinate system (REF system)

```
0  BEGIN PGM 416 MM
1  TOOL CALL "TOUCH_PROBE" Z
2  TCH PROBE 416 PRESET CIRCLE CENTER ~
    Q273=+35            ;CENTER IN 1ST AXIS ~
    Q274=+35            ;CENTER IN 2ND AXIS ~
    Q262=+50            ;NOMINAL DIAMETER ~
    Q291=+90            ;ANGLE OF 1ST HOLE ~
    Q292=+180           ;ANGLE OF 2ND HOLE ~
    Q293=+270           ;ANGLE OF 3RD HOLE ~
    Q261=+15            ;MEASURING HEIGHT ~
    Q260=+10            ;CLEARANCE HEIGHT ~
    Q305=+1             ;NUMBER IN TABLE ~
    Q331=+0             ;PRESET ~
    Q332=+0             ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+7.5           ;1ST CO. FOR TS AXIS ~
    Q383=+7.5           ;2ND CO. FOR TS AXIS ~
    Q384=+20            ;3RD CO. FOR TS AXIS ~
    Q333=+0             ;PRESET ~
    Q320=+0             ;SET-UP CLEARANCE.
3  CYCL DEF 247 PRESETTING ~
    Q339=+1             ;PRESET NUMBER
4  END PGM 416 MM
```

31.4 Touch Probe Cycles: Automatic Workpiece Inspection

31.4.1 Fundamentals

Overview

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.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

The control offers cycles for measuring workpieces automatically:

Cycle		Call	Further information
0	**REF. PLANE** ■ Measuring a coordinate in a selectable axis	**DEF-** active	Page 1778
1	**POLAR PRESET** ■ Measuring a point ■ Probing direction via angle	**DEF-** active	Page 1780
420	**MEASURE ANGLE** ■ Measuring an angle in the working plane	**DEF-** active	Page 1782
421	**MEASURE HOLE** ■ Measuring the position of a hole ■ Measuring the diameter of a hole ■ Nominal-to-actual value comparison, if applicable	**DEF-** active	Page 1785
422	**MEAS. CIRCLE OUTSIDE** ■ Measuring the position of a circular stud ■ Measuring the diameter of a circular stud ■ Nominal-to-actual value comparison, if applicable	**DEF-** active	Page 1791
423	**MEAS. RECTAN. INSIDE** ■ Measuring the position of a rectangular pocket ■ Measuring the length and width of a rectangular pocket ■ Nominal-to-actual value comparison, if applicable	**DEF-** active	Page 1797
424	**MEAS. RECTAN. OUTS.** ■ Measuring the position of a rectangular stud ■ Measuring the length and width of a rectangular stud ■ Nominal-to-actual value comparison, if applicable	**DEF-** active	Page 1802
425	**MEASURE INSIDE WIDTH** ■ Measuring the position of a slot ■ Measuring the width of a slot ■ Nominal-to-actual value comparison, if applicable	**DEF-** active	Page 1806

Cycle		Call	Further information
426	**MEASURE RIDGE WIDTH** ■ Measuring the position of a ridge ■ Measuring the width of a ridge ■ Nominal-to-actual value comparison, if applicable	**DEF**-active	Page 1810
427	**MEASURE COORDINATE** ■ Measuring any coordinate in a selectable axis ■ Nominal-to-actual value comparison, if applicable	**DEF**-active	Page 1814
430	**MEAS. BOLT HOLE CIRC** ■ Measuring the center point of a bolt hole circle ■ Measuring the diameter of a bolt hole circle ■ Nominal-to-actual value comparison, if applicable	**DEF**-active	Page 1819
431	**MEASURE PLANE** ■ Finding the angle of a plane by measuring three points	**DEF**-active	Page 1824

Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles **0** and **1**), you can have the control record the measurement results in a log. In the respective probing cycle you can define if the control is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log to a file, the control by default saves the data as an ASCII file. The control will save the file in the directory that also contains the associated NC program.

The unit of measurement of the main program can be seen in the header of the log file.

> Use the HEIDENHAIN data transfer software TNCremo if you wish to output the measuring log over the data interface.

Example: Measuring log for touch probe cycle **421**:

Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005
Time: 6:55:04
Measuring program: TNC:\GEH35712\CHECK1.H
Type of dimension (0 = MM / 1 = INCH): 0

Nominal values:	
Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000
Given limit values:	
Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000
Minimum limit for center in minor axis:	64.9000
Maximum dimension for hole:	12.0450
Minimum dimension for hole:	12.0000
Actual values:	
Center in reference axis:	50.0810
Center in minor axis:	64.9530
Diameter:	12.0259
Deviations:	
Center in reference axis:	0.0810
Center in minor axis:	-0.0470
Diameter:	0.0259
Further measuring results: Measuring height:	-5.0000

End of measuring log

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. Deviations from the nominal values are saved in parameters **Q161** to **Q166**. Note the table of result parameters listed with every cycle description.

During cycle definition, the control also shows the result parameters for the respective cycle in a help graphic . The highlighted result parameter belongs to that input parameter.

Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters **Q180** to **Q182**.

Parameter value	Measuring status
Q180 = 1	Measurement results are within tolerance
Q181 = 1	Rework is required
Q182 = 1	Scrap

The control sets the rework or scrap marker as soon as one of the measuring values is out of tolerance. To determine which of the measuring results is out of tolerance, check the measuring log, or compare the respective measuring results (**Q150** to **Q160**) with their limit values.

In Cycle **427** the control assumes by default that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.

> The control also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

With most cycles for workpiece inspection, you can have the control perform tolerance monitoring. This requires that you define the necessary limit values during cycle definition. If you do not wish to monitor for tolerances, simply leave the default value 0 for this parameter set this parameter unchanged.

Tool monitoring

With some cycles for workpiece inspection, you can have the control perform tool monitoring. The control then monitors whether

- the tool radius should be compensated due to the deviations from the nominal value (values in **Q16x**)
- the deviations from the nominal value (values in **Q16x**) are greater than the tool breakage tolerance.

Tool compensation

Requirements:

- Active tool table
- Tool monitoring must be switched on in the cycle: Set **Q330** unequal to 0 or enter a tool name. Select the tool name input via **Name** in the action bar.

> - HEIDENHAIN recommends using this function only if the tool to be compensated for is the one that was used to machine the contour as well as if any necessary reworking will also be done with this tool.
> - If you perform several compensation measurements, the control adds the respective measured deviation to the value stored in the tool table.

Milling cutter

If you reference a milling cutter in parameter **Q330**, the appropriate values will be compensates for as follows:

The control always compensates for the tool radius in the **DR** column of the tool table, even if the measured deviation lies within the given tolerance.

You can inquire whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

Turning tool

Only applies to Cycles **421**, **422**, **427**.

If you reference a turning tool in parameter **Q330**, the appropriate values in row DZL and DXL, respectively, will be compensated. The control also monitors the breakage tolerance, which is defined in column LBREAK.

You can poll whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

Compensating for an indexed tool

If you want to automatically compensate the values for an indexed tool with a tool name, program the following:

- **QS0** = "TOOL NAME"
- **FN18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0**= **Q0** +0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool number

Tool breakage monitoring

Requirements:

- Active tool table
- Tool monitoring must be switched on in the cycle (set **Q330** unequal to 0)
- RBREAK must be greater than 0 (in the entered tool number in the table)

  Further information: "Tool data", Page 275

The control will output an error message and stop the program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time, the tool will be deactivated in the tool table (column TL = L).

Reference system for measurement results

The control transfers all measurement results, which reference the active coordinate system, or as the case may be, the shifted or/and rotated/tilted coordinate system, to the result parameters and the log file.

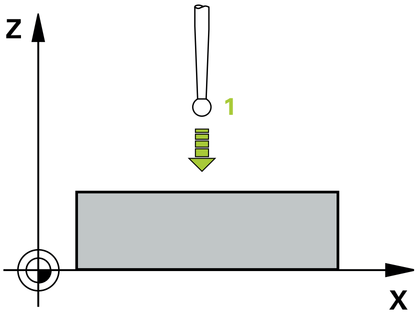
31.4.2 Cycle 0 REF. PLANE

ISO programming
G55

Application

The touch probe cycle measures any position on the workpiece in a selectable axis direction.

Cycle sequence



1 In a 3D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
2 Next, the touch probe performs probing at the probing feed rate (**F** column). The probing direction must be defined in the cycle.
3 After the control has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. In addition, the control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**. For the values in these parameters the control does not account for the stylus length and radius.

Notes

NOTICE
Danger of collision! The control moves the touch probe in a 3D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision! ▶ Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want to assign the coordinate.. Input: **0...1999**
	Probing axis/probing direction? Select the probing axis with the axis key or the alphabetic keyboard, entering the algebraic sign for the probing direction. Input: **-, +**
	Position value? Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input: **-999999999...+999999999**

Example

```
11 TCH PROBE 0.0 REF. PLANE Q9 Z+
12 TCH PROBE 0.1  X+99  Y+22  Z+2
```

31.4.3 Cycle 1 POLAR PRESET

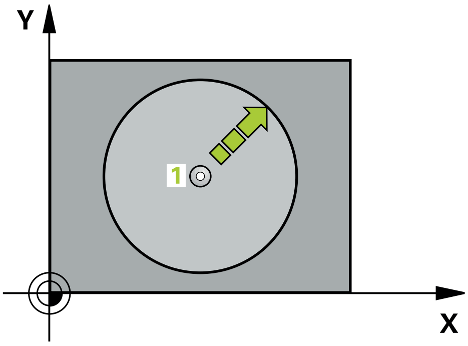
ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **1** measures any position on the workpiece in any probing direction.

Cycle sequence



1 In a 3D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
2 Next, the touch probe performs probing at the probing feed rate (**F** column). During probing, the control moves the touch probe simultaneously in two axes (depending on the probing angle). Use polar angles to define the probing direction in the cycle.
3 After the control has saved the position, the touch probe returns to the starting point. The control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**

Notes

NOTICE

Danger of collision!

The control moves the touch probe in a 3D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The probing axis defined in the cycle specifies the probing plane:
  Probing axis X: X/Y plane
  Probing axis Y: Y/Z plane
  Probing axis Z: Z/X plane

Cycle parameters

Help graphic	Parameter
	Probing axis? Enter the probing axis with the axis key or the alphabetic keyboard. Confirm with the **ENT** key. Input: **X**, **Y**, or **Z**
	Probing angle? Angle measured from the probing axis in which the touch probe will move. Input: **-180...+180**
	Position value? Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input: **-999999999...+999999999**

Example

```
11 TCH PROBE 1.0 POLAR PRESET
12 TCH PROBE 1.1 X WINKEL:+30
13 TCH PROBE 1.2  X+0  Y+10  Z+3
```

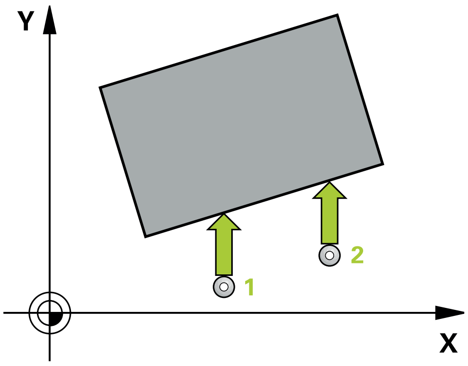
31.4.4 Cycle 420 MEASURE ANGLE

ISO programming
G420

Application

Touch probe cycle **420** measures the angle that any straight line on the workpiece forms with the main axis of the working plane.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account for probe movements in any probing direction. When the probe movement starts, the center of the ball tip will be offset by this sum in the direction opposite the probing direction.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

Q parameter number	Meaning
Q150	The measured angle is referenced to the reference axis of the working plane.

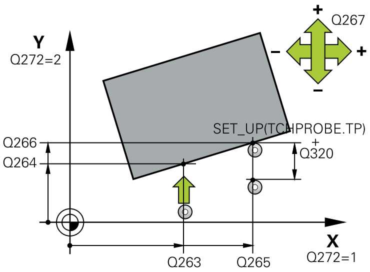
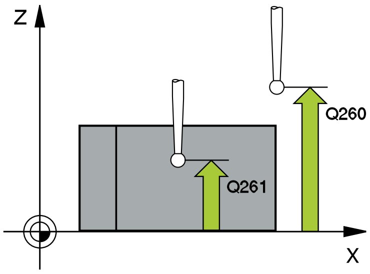
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:
  - If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
  - If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q272 Meas. axis (1/2/3, 1=ref. axis)? Axis in which the measurement will be made: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis **3**: Touch probe axis = measuring axis Input: **1**, **2**, **3**
	Q267 Trav. direction 1 (+1=+ / -1=-)? Direction in which the touch probe will approach the workpiece: **-1**: Negative traverse direction **+1**: Positive traverse direction Input: **-1**, **+1**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of **Q320**, **SET_UP**, and the ball-tip radius, even when probing in the tool axis direction. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: Define whether the control will create a measuring log: **1**: Create a measuring log: The control will save the **log file named TCHPR420.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with **NC Start**) Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 420 MEASURE ANGLE ~
    Q263=+10            ;1ST POINT 1ST AXIS ~
    Q264=+10            ;1ST POINT 2ND AXIS ~
    Q265=+15            ;2ND PNT IN 1ST AXIS ~
    Q266=+95            ;2ND POINT 2ND AXIS ~
    Q272=+1             ;MEASURING AXIS ~
    Q267=-1             ;TRAVERSE DIRECTION ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+10            ;CLEARANCE HEIGHT ~
    Q301=+1             ;MOVE TO CLEARANCE ~
    Q281=+1             ;MEASURING LOG
```

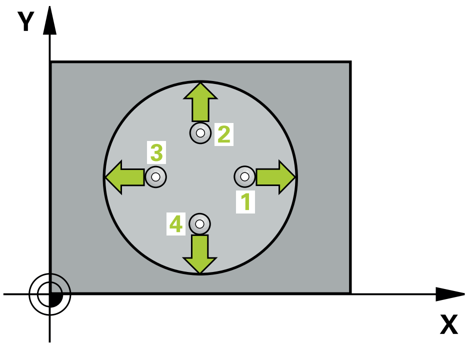
31.4.5 Cycle 421 MEASURE HOLE

ISO programming
G421

Application

Touch probe cycle **421** measures the center point and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the SET_UP column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

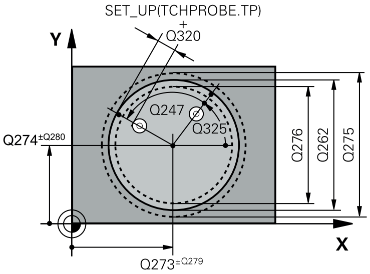
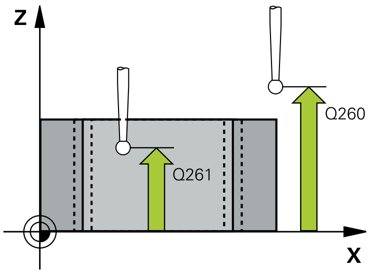
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The nominal diameter **Q262** must be between the minimum and maximum dimension (**Q276**/**Q275**).
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter Q330 references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Center of the hole in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Center of the hole in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Enter the diameter of the hole. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q275 Maximum limit of size for hole? Maximum permissible diameter for the hole (circular pocket) Input: **0...99999.9999**
	Q276 Minimum limit of size? Minimum permissible diameter for the hole (circular pocket) Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR421.TXT** by default in the directory that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring : **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776

Help graphic	Parameter
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**
	Q498 Reverse tool (0=no/1=yes)? Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following: **1**: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1 **0**: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example , Cycle **800** and parameter **Reverse the tool Q498** = 0 Input: **0**, **1**
	Q531 Angle of incidence? Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**). Input: **-180...+180**

Example

```
11 TCH PROBE 421 MEASURE HOLE ~
    Q273=+50            ;CENTER IN 1ST AXIS ~
    Q274=+50            ;CENTER IN 2ND AXIS ~
    Q262=+15.25         ;NOMINAL DIAMETER ~
    Q325=+0             ;STARTING ANGLE ~
    Q247=+60            ;STEPPING ANGLE ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+1             ;MOVE TO CLEARANCE ~
    Q275=+15.34         ;MAXIMUM LIMIT ~
    Q276=+15.16         ;MINIMUM LIMIT ~
    Q279=+0.1           ;TOLERANCE 1ST CENTER ~
    Q280=+0.1           ;TOLERANCE 2ND CENTER ~
    Q281=+1             ;MEASURING LOG ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL ~
    Q423=+4             ;NO. OF PROBE POINTS ~
    Q365=+1             ;TYPE OF TRAVERSE ~
    Q498=+0             ;REVERSE TOOL ~
    Q531=+0             ;ANGLE OF INCIDENCE
```

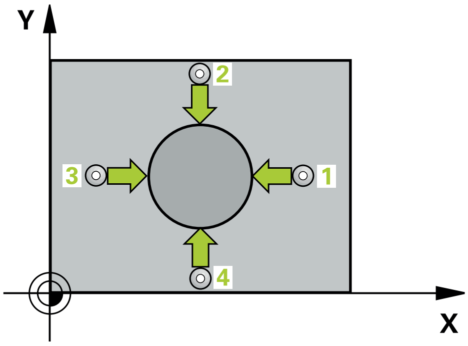
31.4.6 Cycle 422 MEAS. CIRCLE OUTSIDE

ISO programming
G422

Application

Touch probe cycle **422** measures the center point and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

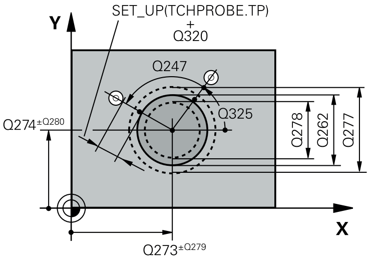
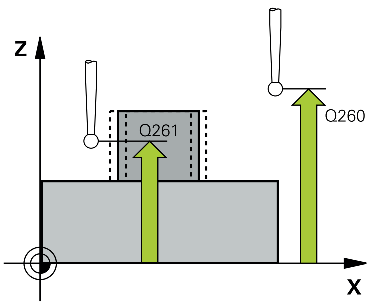
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter Q330 references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Center of the stud in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Center of the stud in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Enter the diameter of the stud. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the machining direction (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q277 Maximum limit of size for stud? Maximum permissible diameter for the stud. Input: **0...99999.9999**
	Q278 Minimum limit of size for stud? Minimum permissible diameter for the stud. Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR422.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring: **0**: Monitoring not active **> 0**: Tool number in tool table TOOL.T Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**

Help graphic	Parameter
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**
	Q498 Reverse tool (0=no/1=yes)? Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following: **1**: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1 **0**: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example , Cycle **800** and parameter **Reverse the tool Q498** = 0 Input: **0**, **1**
	Q531 Angle of incidence? Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**). Input: **-180...+180**

Example

```
11 TCH PROBE 422 MEAS. CIRCLE OUTSIDE ~
    Q273=+50                ;CENTER IN 1ST AXIS ~
    Q274=+50                ;CENTER IN 2ND AXIS ~
    Q262=+75                ;NOMINAL DIAMETER ~
    Q325=+90                ;STARTING ANGLE ~
    Q247=+30                ;STEPPING ANGLE ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+10                ;CLEARANCE HEIGHT ~
    Q301=+0                 ;MOVE TO CLEARANCE ~
    Q277=+35.15             ;MAXIMUM LIMIT ~
    Q278=+34.9              ;MINIMUM LIMIT ~
    Q279=+0.05              ;TOLERANCE 1ST CENTER ~
    Q280=+0.05              ;TOLERANCE 2ND CENTER ~
    Q281=+1                 ;MEASURING LOG ~
    Q309=+0                 ;PGM STOP TOLERANCE ~
    Q330=+0                 ;TOOL ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q365=+1                 ;TYPE OF TRAVERSE ~
    Q498=+0                 ;REVERSE TOOL ~
    Q531=+0                 ;ANGLE OF INCIDENCE
```

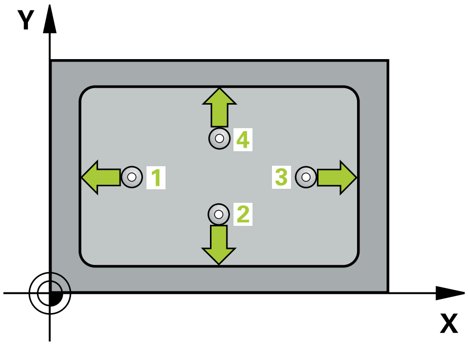
31.4.7 Cycle 423 MEAS. RECTAN. INSIDE

ISO programming
G423

Application

Touch probe cycle **423** finds the center, length, and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

Notes

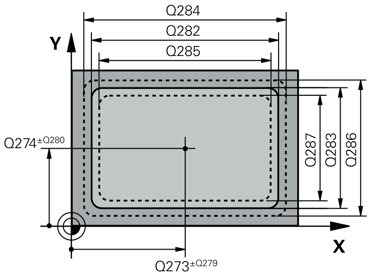
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter

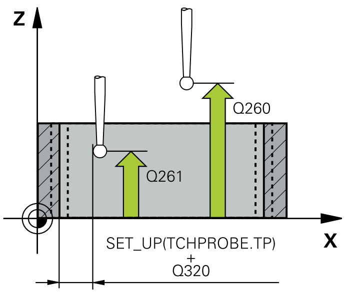


Q273 Center in 1st axis (nom. value)?
Center of the pocket in the main axis of the working plane. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?
Center of the pocket in the secondary axis of the working plane. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q282 1st side length (nominal value)?
Pocket length, parallel to the main axis of the working plane
Input: **0...99999.9999**

Q283 2nd side length (nominal value)?
Pocket length, parallel to the secondary axis of the working plane
Input: **0...99999.9999**



Q261 Measuring height in probe axis?
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

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** or **PREDEF**

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** or **PREDEF**

Q301 Move to clearance height (0/1)?
Specify how the touch probe moves between measuring points:
0: Move at measuring height between measuring points
1: Move at clearance height between measuring points
Input: **0**, **1**

Q284 Max. size limit 1st side length?
Maximum permissible length for the pocket
Input: **0...99999.9999**

Q285 Min. size limit 1st side length?
Minimum permissible length for the pocket
Input: **0...99999.9999**

Help graphic	Parameter
	Q286 Max. size limit 2nd side length? Maximum permissible width for the pocket Input: **0...99999.9999**
	Q287 Min. size limit 2nd side length? Minimum permissible width for the pocket Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log. **1**: Create a measuring log: The control will save the **log file named TCHPR423.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen.Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring: **0**: Monitoring not active **> 0**: Tool number in tool table TOOL.T Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776

Example

```
11 TCH PROBE 423 MEAS. RECTAN. INSIDE ~
    Q273=+50            ;CENTER IN 1ST AXIS ~
    Q274=+50            ;CENTER IN 2ND AXIS ~
    Q282=+80            ;FIRST SIDE LENGTH ~
    Q283=+60            ;2ND SIDE LENGTH ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+10            ;CLEARANCE HEIGHT ~
    Q301=+1             ;MOVE TO CLEARANCE ~
    Q284=+0             ;MAX. LIMIT 1ST SIDE ~
    Q285=+0             ;MIN. LIMIT 1ST SIDE ~
    Q286=+0             ;MAX. LIMIT 2ND SIDE ~
    Q287=+0             ;MIN. LIMIT 2ND SIDE ~
    Q279=+0             ;TOLERANCE 1ST CENTER ~
    Q280=+0             ;TOLERANCE 2ND CENTER ~
    Q281=+1             ;MEASURING LOG ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL
```

31.4.8 Cycle 424 MEAS. RECTAN. OUTS.

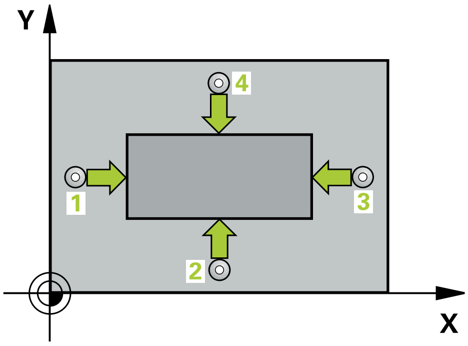
ISO programming

G424

Application

Touch probe cycle **424** finds the center, length, and width of a rectangular stud.
If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

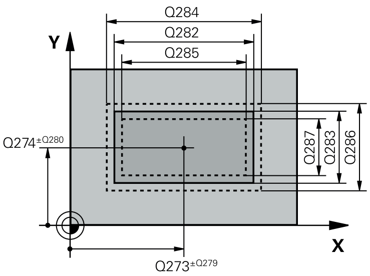
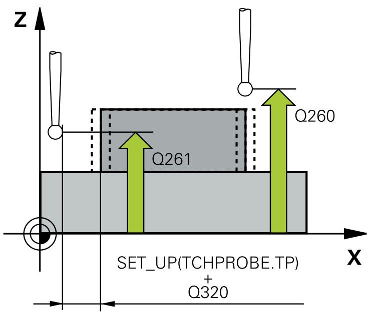
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Center of the stud in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Center of the stud in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q282 1st side length (nominal value)? Length of stud parallel to the main axis of the working plane Input: **0...99999.9999**
	Q283 2nd side length (nominal value)? Length of stud parallel to the secondary axis of the working plane Input: **0...99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q284 Max. size limit 1st side length? Maximum permissible length for the stud Input: **0...99999.9999**

Help graphic	Parameter
	Q285 Min. size limit 1st side length? Minimum permissible length for the stud Input: **0...99999.9999**
	Q286 Max. size limit 2nd side length? Maximum permissible width for the stud Input: **0...99999.9999**
	Q287 Min. size limit 2nd side length? Minimum permissible width for the stud Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR424.TXT** in the folder that also contains the .h file **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring : **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776

Example

```
11 TCH PROBE 424 MEAS. RECTAN. OUTS. ~
    Q273=+50            ;CENTER IN 1ST AXIS ~
    Q274=+50            ;2ND CENTER 2ND AXIS ~
    Q282=+75            ;FIRST SIDE LENGTH ~
    Q283=+35            ;2ND SIDE LENGTH ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q284=+75.1          ;MAX. LIMIT 1ST SIDE ~
    Q285=+74.9          ;MIN. LIMIT 1ST SIDE ~
    Q286=+35            ;MAX. LIMIT 2ND SIDE ~
    Q287=+34.95         ;MIN. LIMIT 2ND SIDE ~
    Q279=+0.1           ;TOLERANCE 1ST CENTER ~
    Q280=+0.1           ;TOLERANCE 2ND CENTER ~
    Q281=+1             ;MEASURING LOG ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL
```

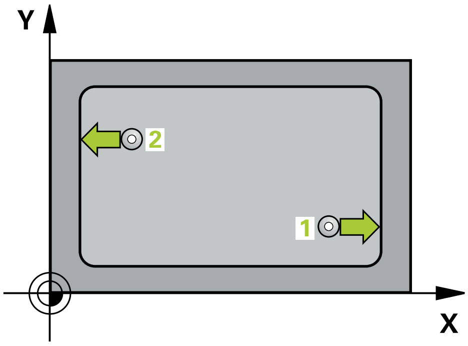
31.4.9 Cycle 425 MEASURE INSIDE WIDTH

ISO programming
G425

Application

Touch probe cycle **425** measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation value in a Q parameter.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the positive direction of the programmed axis.
3 If you enter an offset for the second measurement, the control then moves the touch probe (if required, at clearance height) to the next touch point **2** and probes that point. If the nominal length is large, the control moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the control measures the width in the exact opposite direction.
4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

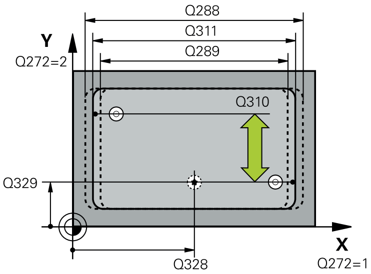
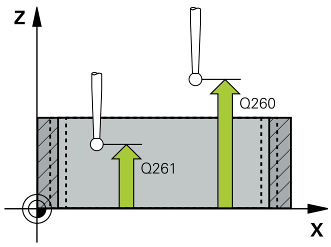
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The nominal length **Q311** must be between the minimum and maximum dimension (**Q276**/**Q275**).

Cycle parameters

Help graphic	Parameter
	**Q328 Starting point in 1st axis?** Starting point for probing in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q329 Starting point in 2nd axis? Starting point for probing in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q310 Offset for 2nd measuremnt (+/-)? Distance by which the touch probe is offset before the second measurement. If you enter 0, the control does not offset the touch probe. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis Input: **1**, **2**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+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** or **PREDEF**
	Q311 Nominal length? Nominal value of the length to be measured Input: **0...99999.9999**
	Q288 Maximum limit of size? Maximum permissible length Input: **0...99999.9999**
	Q289 Minimum limit of size? Minimum permissible length Input: **0...99999.9999**

Help graphic	Parameter
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR425.TXT** in the folder that also contains the .h file **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring : **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**

Example

```
11 TCH PROBE 425 MEASURE INSIDE WIDTH ~
    Q328=+75            ;STARTNG PNT 1ST AXIS ~
    Q329=-12.5          ;STARTNG PNT 2ND AXIS ~
    Q310=+0             ;OFFS. 2ND MEASUREMNT ~
    Q272=+1             ;MEASURING AXIS ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q260=+10            ;CLEARANCE HEIGHT ~
    Q311=+25            ;NOMINAL LENGTH ~
    Q288=+25.05         ;MAXIMUM LIMIT ~
    Q289=+25            ;MINIMUM LIMIT ~
    Q281=+1             ;MEASURING LOG ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q301=+0             ;MOVE TO CLEARANCE
```

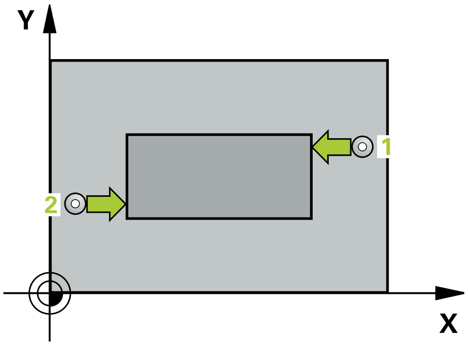
31.4.10 Cycle 426 MEASURE RIDGE WIDTH

ISO programming
G426

Application

Touch probe cycle **426** measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1596
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the negative direction of the programmed axis.
3 Then the touch probe moves at clearance height to the next touch point and probes it.
4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

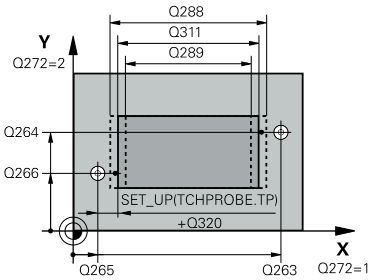
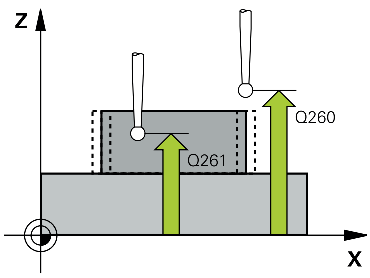
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis Input: **1**, **2**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**
	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** or **PREDEF**
	Q311 Nominal length? Nominal value of the length to be measured Input: **0...99999.9999**
	Q288 Maximum limit of size? Maximum permissible length Input: **0...99999.9999**

Help graphic	Parameter
	Q289 Minimum limit of size? Minimum permissible length Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR426.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Q330 Define whether the control should perform tool monitoring : **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776

Example

```
11 TCH PROBE 426 MEASURE RIDGE WIDTH ~
    Q263=+50            ;1ST POINT 1ST AXIS ~
    Q264=+25            ;1ST POINT 2ND AXIS ~
    Q265=+50            ;2ND PNT IN 1ST AXIS ~
    Q266=+85            ;2ND PNT IN 2ND AXIS ~
    Q272=+2             ;MEASURING AXIS ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q311=+45            ;NOMINAL LENGTH ~
    Q288=+45            ;MAXIMUM LIMIT ~
    Q289=+44.95         ;MINIMUM LIMIT ~
    Q281=+1             ;MEASURING LOG ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL
```

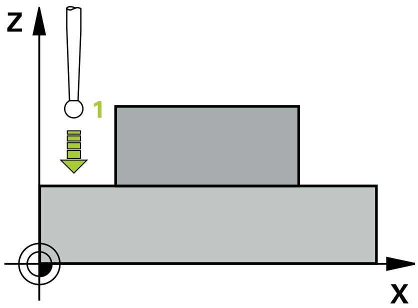
31.4.11 Cycle 427 MEASURE COORDINATE

ISO programming
G427

Application

Touch probe cycle **427** measures a coordinate in a selectable axis and saves the value in a Q parameter. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

   Further information: "Positioning logic", Page 1596
2 Then the control positions the touch probe to the specified touch point **1** in the working plane and measures the actual value in the selected axis.
3 Finally, the control returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

Q parameter number	Meaning
Q160	Measured coordinate

Notes

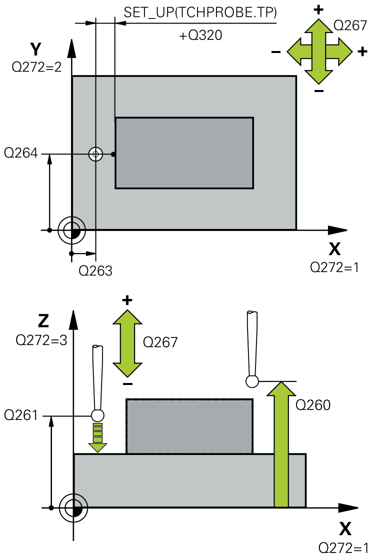
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).
- If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The measuring height **Q261** must be between the minimum and maximum dimension (**Q276**/**Q275**).
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter Q330 references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic	Parameter



Q263 1st measuring point in 1st axis?
Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?
Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

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** or **PREDEF**

Q272 Meas. axis (1/2/3, 1=ref. axis)?
Axis in which the measurement will be made:
1: Main axis = measuring axis
2: Secondary axis = measuring axis
3: Touch probe axis = measuring axis
Input: **1**, **2**, **3**

Q267 Trav. direction 1 (+1=+ / -1=-)?
Direction in which the touch probe will approach the workpiece:
-1: Negative traverse direction
+1: Positive traverse direction
Input: **-1**, **+1**

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** or **PREDEF**

Help graphic	Parameter
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR427.TXT** in the folder that also contains the associated NC program. **2:** Interrupt the program run and display the measuring log on the control screen.Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q288 Maximum limit of size? Maximum permissible value Input: **-99999.9999...+99999.9999**
	Q289 Minimum limit of size? Minimum permissible value Input: **-99999.9999...+99999.9999**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring : **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776

Help graphic	Parameter
	Q498 Reverse tool (0=no/1=yes)? Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following: **1**: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1 **0**: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example , Cycle **800** and parameter **Reverse the tool Q498** = 0 Input: **0**, **1**
	Q531 Angle of incidence? Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**). Input: **-180...+180**

Example

```
11 TCH PROBE 427 MEASURE COORDINATE ~
    Q263=+35            ;1ST POINT 1ST AXIS ~
    Q264=+45            ;1ST POINT 2ND AXIS ~
    Q261=+5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q272=+3             ;MEASURING AXIS ~
    Q267=-1             ;TRAVERSE DIRECTION ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q281=+1             ;MEASURING LOG ~
    Q288=+5.1           ;MAXIMUM LIMIT ~
    Q289=+4.95          ;MINIMUM LIMIT ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL ~
    Q498=+0             ;REVERSE TOOL ~
    Q531=+0             ;ANGLE OF INCIDENCE
```

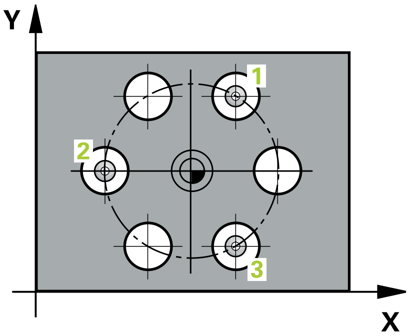
31.4.12 Cycle 430 MEAS. BOLT HOLE CIRC

ISO programming
G430

Application

Touch probe cycle **430** finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.

   Further information: "Positioning logic", Page 1596
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
7 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation of bolt circle diameter

Notes

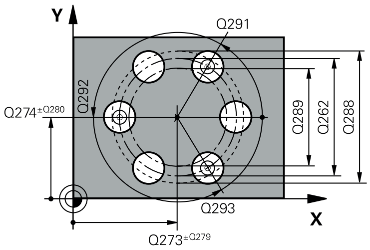
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **430** only monitors for tool breakage; there is no automatic tool compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic | **Parameter**



Q273 Center in 1st axis (nom. value)?
Bolt hole circle center (nominal value) in the main axis of the working plane. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

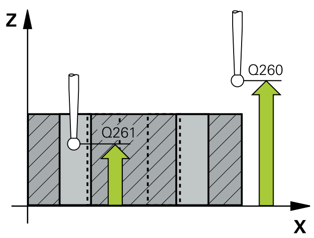
Q274 Center in 2nd axis (nom. value)?
Bolt hole circle center (nominal value) in the secondary axis of the working plane. This value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?
Enter the diameter of the hole.
Input: **0...99999.9999**

Q291 Polar coord. angle of 1st hole?
Polar coordinate angle of the first hole center in the working plane. This value has an absolute effect.
Input: **-360.000...+360.000**

Q292 Polar coord. angle of 2nd hole?
Polar coordinate angle of the second hole center in the working plane. This value has an absolute effect.
Input: **-360.000...+360.000**

Q293 Polar coord. angle of 3rd hole?
Polar coordinate angle of the third hole center in the working plane. This value has an absolute effect.
Input: **-360.000...+360.000**



Q261 Measuring height in probe axis?
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.
Input: **-99999.9999...+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** or **PREDEF**

Q288 Maximum limit of size?
Maximum permissible diameter of bolt hole circle
Input: **0...99999.9999**

Q289 Minimum limit of size?
Minimum permissible diameter of bolt hole circle
Input: **0...99999.9999**

Q279 Tolerance for center 1st axis?
Permissible position deviation in the main axis of the working plane.
Input: **0...99999.9999**

Help graphic	Parameter
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR430.TXT** in the folder that also contains the associated NC program **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring : **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters **Further information:** "Tool monitoring", Page 1776

Example

```
11 TCH PROBE 430 MEAS. BOLT HOLE CIRC ~
    Q273=+50            ;CENTER IN 1ST AXIS ~
    Q274=+50            ;CENTER IN 2ND AXIS ~
    Q262=+80            ;NOMINAL DIAMETER ~
    Q291=+0             ;ANGLE OF 1ST HOLE ~
    Q292=+90            ;ANGLE OF 2ND HOLE ~
    Q293=+180           ;ANGLE OF 3RD HOLE ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q260=+10            ;CLEARANCE HEIGHT ~
    Q288=+80.1          ;MAXIMUM LIMIT ~
    Q289=+79.9          ;MINIMUM LIMIT ~
    Q279=+0.15          ;TOLERANCE 1ST CENTER ~
    Q280=+0.15          ;TOLERANCE 2ND CENTER ~
    Q281=+1             ;MEASURING LOG ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL
```

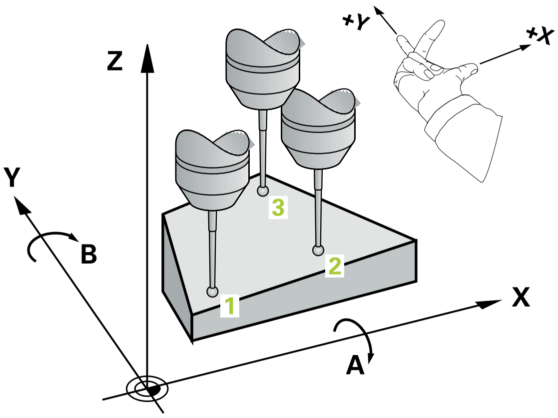
31.4.13 Cycle 431 MEASURE PLANE

ISO programming
G431

Application

Touch probe cycle **431** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) at the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the set-up clearance in the direction opposite to the direction of probing.
 Further information: "Positioning logic", Page 1596
2 The touch probe returns to the clearance height and then moves in the working plane to touch point **2** and measures the actual value of the second touch point in the plane.
3 The touch probe returns to the clearance height and then moves in the working plane to touch point **3** and measures the actual value of the third touch point in the plane.
4 Finally the control returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

Q parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to **Q175**	Measured values in the touch probe axis (first to third measurement)

Notes

NOTICE

Risk of collision!

If you save the angle values in the preset table and then tilt the tool by programming **PLANE SPATIAL** with **SPA** = 0; **SPB** = 0; **SPC** = 0, there are multiple solutions in which the tilting axes are at 0. There is a risk of collision!

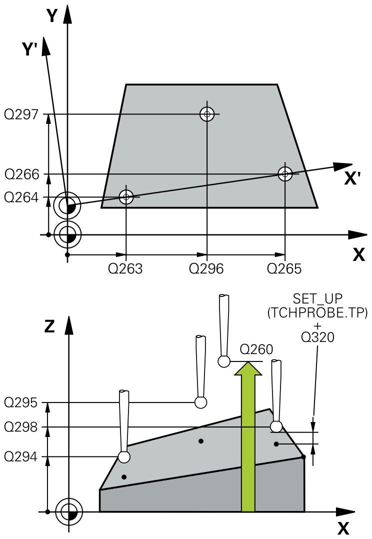
- Make sure to program **SYM** (**SEQ**) + or **SYM** (**SEQ**) -

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three measuring points are not positioned on a straight line.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The spatial angles that are needed for the **Tilt working plane**function are saved in parameters **Q170** to **Q172**. With the first two measuring points, you also specify the direction of the main axis when tilting the working plane.
- The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q295 2nd measuring point in 3rd axis? Coordinate of the second touch point in the touch probe axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q296 3rd measuring point in 1st axis? Coordinate of the third touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q297 3rd measuring point in 2nd axis? Coordinate of the third touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q298 3rd measuring point in 3rd axis? Coordinate of the third touch point in the touch probe axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	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** or **PREDEF**

Help graphic	Parameter
	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** or **PREDEF**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the folder that also contains the associated NC program **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**

Example

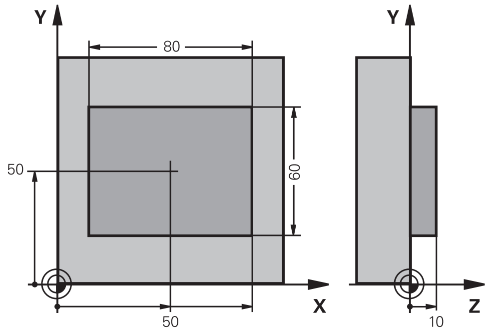
```
11 TCH PROBE 431 MEASURE PLANE ~
    Q263=+20            ;1ST POINT 1ST AXIS ~
    Q264=+20            ;1ST POINT 2ND AXIS ~
    Q294=-10            ;1ST POINT 3RD AXIS ~
    Q265=+50            ;2ND PNT IN 1ST AXIS ~
    Q266=+80            ;2ND PNT IN 2ND AXIS ~
    Q295=+0             ;2ND PNT IN 3RD AXIS ~
    Q296=+90            ;3RD PNT IN 1ST AXIS ~
    Q297=+35            ;THIRD POINT 2ND AXIS ~
    Q298=+12            ;3RD PNT IN 3RD AXIS ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+5             ;CLEARANCE HEIGHT ~
    Q281=+1             ;MEASURING LOG
```

31.4.14 Programming Examples

Example: Measuring and reworking a rectangular stud

Program sequence

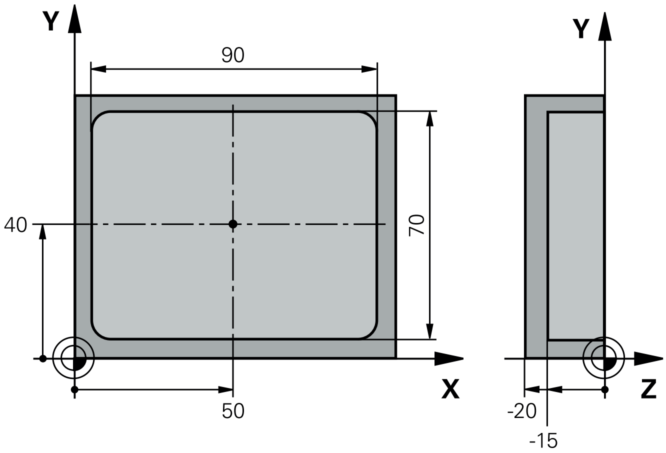
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 5 Z S6000	; Tool call: roughing
2 Q1 = 81	; Rectangle length in X (roughing dimension)
3 Q2 = 61	; Rectangle length in Y (roughing dimension)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CALL LBL 1	; Call the subprogram for machining
6 L Z+100 R0 FMAX	; Retract the tool
7 TOOL CALL 600 Z	; Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS. ~	
Q273=+50 ;CENTER IN 1ST AXIS ~	
Q274=+50 ;CENTER IN 2ND AXIS ~	
Q282=+80 ;FIRST SIDE LENGTH ~	
Q283=+60 ;2ND SIDE LENGTH ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q320=+0 ;SET-UP CLEARANCE ~	
Q260=+30 ;CLEARANCE HEIGHT ~	
Q301=+0 ;MOVE TO CLEARANCE ~	
Q284=+0 ;MAX. LIMIT 1ST SIDE ~	
Q285=+0 ;MIN. LIMIT 1ST SIDE ~	
Q286=+0 ;MAX. LIMIT 2ND SIDE ~	
Q287=+0 ;MIN. LIMIT 2ND SIDE ~	
Q279=+0 ;TOLERANCE 1ST CENTER ~	
Q280=+0 ;TOLERANCE 2ND CENTER ~	
Q281=+0 ;MEASURING LOG ~	
Q309=+0 ;PGM STOP TOLERANCE ~	
Q330=+0 ;TOOL	

9 Q1 = Q1 - Q164	; Calculate the length in X based on the measured deviation
10 Q2 = Q2 - Q165	; Calculate the length in Y based on the measured deviation
11 L Z+100 R0 FMAX	; Retract the touch probe
12 TOOL CALL 25 Z S8000	; Tool call: finishing
13 L Z+100 R0 FMAX M3	; Retract the tool, end of program
14 CALL LBL 1	; Call the subprogram for machining
15 L Z+100 R0 FMAX	
16 M30	
17 LBL 1	; Subprogram with rectangular stud machining cycle
18 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+Q1 ;FIRST SIDE LENGTH ~	
Q424=+82 ;WORKPC. BLANK SIDE 1 ~	
Q219=+Q2 ;2ND SIDE LENGTH ~	
Q425=+62 ;WORKPC. BLANK SIDE 2 ~	
Q220=+0 ;RADIUS / CHAMFER ~	
Q368=+0.1 ;ALLOWANCE FOR SIDE ~	
Q224=+0 ;ANGLE OF ROTATION ~	
Q367=+0 ;STUD POSITION ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-10 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q206=+3000 ;FEED RATE FOR PLNGNG ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+10 ;SURFACE COORDINATE ~	
Q204=+20 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q437=+0 ;APPROACH POSITION ~	
Q215=+0 ;MACHINING OPERATION ~	
Q369=+0 ;ALLOWANCE FOR FLOOR ~	
Q338=+20 ;INFEED FOR FINISHING ~	
Q385=+500 ;FINISHING FEED RATE	
19 L X+50 Y+50 R0 FMAX M99	; Cycle call
20 LBL 0	; End of subprogram
21 END PGM TOUCHPROBE MM	

Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM TOUCHPROBE_2 MM	
1 TOOL CALL 600 Z	; Tool call: touch probe
2 L Z+100 R0 FMAX	; Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50 ;CENTER IN 1ST AXIS ~	
Q274=+40 ;CENTER IN 2ND AXIS ~	
Q282=+90 ;FIRST SIDE LENGTH ~	
Q283=+70 ;2ND SIDE LENGTH ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q320=+2 ;SET-UP CLEARANCE ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q301=+0 ;MOVE TO CLEARANCE ~	
Q284=+90.15 ;MAX. LIMIT 1ST SIDE ~	
Q285=+89.95 ;MIN. LIMIT 1ST SIDE ~	
Q286=+70.1 ;MAX. LIMIT 2ND SIDE ~	
Q287=+69.9 ;MIN. LIMIT 2ND SIDE ~	
Q279=+0.15 ;TOLERANCE 1ST CENTER ~	
Q280=+0.1 ;TOLERANCE 2ND CENTER ~	
Q281=+1 ;MEASURING LOG ~	
Q309=+0 ;PGM STOP TOLERANCE ~	
Q330=+0 ;TOOL	
4 L Z+100 R0 FMAX	; Retract the tool, end of program
5 M30	
6 END PGM TOUCHPROBE_2 MM	

31.5 Touch Probe Cycles: Special Functions

31.5.1 Fundamentals

Overview

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.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

The control provides cycles for the following special purposes:

Cycle		Call	Further information
3	**MEASURING** ■ Touch probe cycle for defining OEM cycles	**DEF**-active	Page 1832
4	**MEASURING IN 3-D** ■ Measuring any position	**DEF**-active	Page 1834
444	**PROBING IN 3-D** ■ Measuring any position ■ Determining the deviation from the nominal coordinates	**DEF**-active	Page 1837
441	**FAST PROBING** ■ Touch probe cycle for defining various touch probe parameters	**DEF**-active	Page 1843
1493	**EXTRUSION PROBING** ■ Touch probe cycle for defining an extrusion ■ Extrusion direction, length, and number of extrusion points can be programmed	**DEF**-active	Page 1845

31.5.2 Cycle 3 MEASURING

ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **3** measures any position on the workpiece in a selectable probing direction. Unlike other touch probe cycles, Cycle **3** enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by a definable value **MB** after determining the measured value.

Cycle sequence

1 The touch probe moves from the current position at the specified feed rate in the defined probing direction. Use polar angles to define the probing direction in the cycle.
2 After the control has saved the position, the touch probe stops. The control saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The control does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.
3 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Notes

⚙	The exact behavior of touch probe cycle **3** is defined by your machine manufacturer or a software manufacturer who uses it within specific touch probe cycles.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other touch probe cycles, do not apply in touch probe cycle **3**.
- Remember that the control always writes to four successive Q parameters.
- If the control was not able to determine a valid touch point, the NC program is run without an error message. In this case the control assigns the value –1 to the fourth result parameter so that you can deal with the error yourself.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.

i	With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle sequences through the probe input X12 or X13.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input: **0...1999**
	Probing axis? Enter the axis in whose direction the touch probe will move and confirm with the **ENT** key. Input: **X**, **Y**, or **Z**
	Probing angle? This angle defines the probing direction. The angle refers to the probe axis. Confirm with the **ENT** key. Input: **-180...+180**
	Maximum measuring range? Enter the maximum distance from the starting point by which the touch probe will move. Confirm with ENT. Input: **0...999999999**
	Feed rate measurement Enter the measuring feed rate in mm/min. Input: **0...3000**
	Maximum retraction distance? Traverse path in the direction opposite to the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision. Input: **0...999999999**
	Reference system? (0=ACT/1=REF) Define whether the probing direction and measurement result will be referenced to the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**): **0**: Perform the probing operation in the current system and save the measurement result in the **ACT** system **1**: Perform the probing operation in the machine-based REF system. Save the measurement result in the REF system. Input: **0**, **1**

Help graphic	Parameter
	Error mode? (0=OFF/1=ON) Define whether the control will issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the control saves the value **-1** in the 4th result parameter and continues the cycle: **0**: Issue error message **1**: Do not issue error message Input: **0**, **1**

Example

```
11 TCH PROBE 3.0 MEASURING
12 TCH PROBE 3.1 Q1
13 TCH PROBE 3.2 X ANGLE:+15
14 TCH PROBE 3.3 ABST+10 F100 MB1 REFERENCE SYSTEM:0
15 TCH PROBE 3.4 ERRORMODE1
```

31.5.3 Cycle 4 MEASURING IN 3-D

ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **4** measures any position on the workpiece in the probing direction defined by a vector. Unlike other touch probe cycles, Cycle **4** enables you to enter the probing distance and probing feed rate directly. You can also define the distance by which the touch probe retracts after acquiring the probed value.

Cycle **4** is an auxiliary cycle that can be used for probing with any touch probe (TS or TT). The control does not provide a cycle for calibrating the TS touch probe in any probing direction.

Cycle sequence

1 The control moves the touch probe from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).
2 After the control has saved the position, the control stops the probe movement. The control saves the X, Y, Z coordinates of the probing position in three successive Q parameters. You define the number of the first parameter in the cycle. If you are using a TS touch probe, the probe result is corrected by the calibrated center offset.
3 Finally, the control retracts the touch probe in the direction opposite to the direction of probing. You define the traverse distance in parameter **MB**—the touch probe is moved to a point no farther than the starting point.

> Ensure during pre-positioning that the control moves the probe-tip center without compensation to the defined position.

Notes

NOTICE
Danger of collision! If the control was not able to determine a valid touch point, the 4th result parameter will have the value –1. The control does **not** interrupt the program run! There is a danger of collision! ▶ Make sure that all touch points can be reached.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.
- Remember that the control always writes to four successive Q parameters.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input: **0...1999**
	Relative measuring path in X? X component of the direction vector defining the direction in which the touch probe will move. Input: **-999999999...+999999999**
	Relative measuring path in Y? Y component of the direction vector defining the direction in which the touch probe will move. Input: **-999999999...+999999999**
	Relative measuring path in Z? Z component of the direction vector defining the direction in which the touch probe will move. Input: **-999999999...+999999999**
	Maximum measuring range? Enter the maximum distance from the starting point by which the touch probe will move along the direction vector. Input: **-999999999...+999999999**
	Feed rate measurement Enter the measuring feed rate in mm/min. Input: **0...3000**
	Maximum retraction distance? Traverse path in the direction opposite the probing direction, after the stylus was deflected. Input: **0...999999999**
	Reference system? (0=ACT/1=REF) Define whether the result of probing will be saved in the input coordinate system (**ACT**), or with respect to the machine coordinate system (**REF**): **0**: Save the measurement result in the **ACT** system **1**: Save the measurement result in the **REF** system Input: **0**, **1**

Example

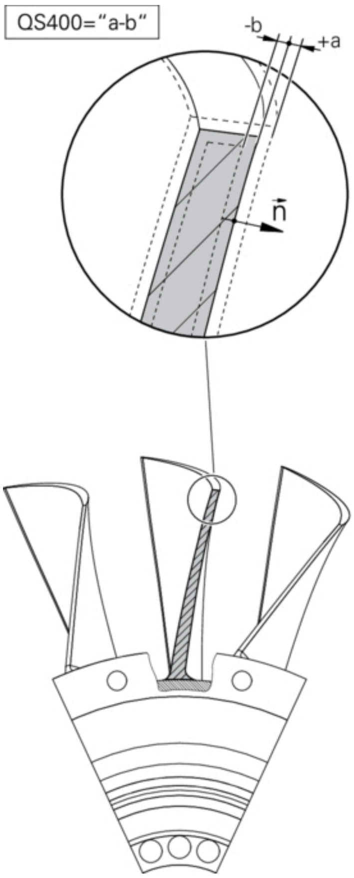
```
11 TCH PROBE 4.0 MEASURING IN 3-D
12 TCH PROBE 4.1 Q1
13 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
14 TCH PROBE 4.3 ABST+45 F100 MB50 REFERENCE SYSTEM:0
```

31.5.4 Cycle 444 PROBING IN 3-D

ISO programming
G444

Application

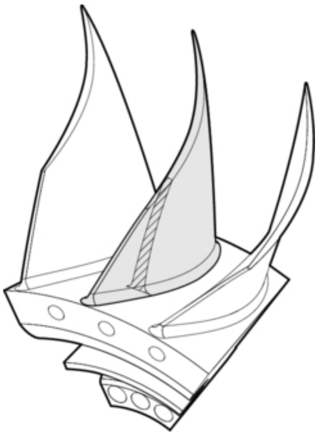
⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



Cycle **444** checks one specific point on the surface of a part. This cycle is used, for example, to measure free-form surfaces of moldmaking parts. It can be determined whether a point on the surface of the part lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.

Cycle **444** probes any point in three dimensions and determines the deviation from a nominal coordinate. A normal vector, defined in parameters **Q581**, **Q582**, and **Q583**, is used for this purpose. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range **QS400** defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the control outputs a log and the deviations are stored in the Q parameters listed below.

Cycle sequence



1 Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate: Distance = ball-tip radius + **SET_UP** value from the tchprobe.tp table (TNC:\table \tchprobe.tp) + **Q320**. Pre-positioning takes a clearance height into account.

   Further information: "Executing touch probe cycles", Page 1596
2 The touch probe then approaches the nominal coordinate. The probing distance is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.
3 After the control has saved the position, the touch probe is retracted and stopped. The control saves the measured coordinates of the contact point in Q parameters.
4 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Result parameters

The control stores the probing results in the following parameters:

Q parameter number	Meaning
Q151	Measured position in main axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in main axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3D deviation ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

Log function

Once probing has finished, the control generates a log in HTML format. The log includes the results from the main, secondary, and tool axes as well as the 3D deviation. The control saves the log in the same folder in which the *.h file is located (as long as no path has been configured for FN16).

The log contains the following data on the main, secondary, and tool axes:

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- If a tolerance **QS400** was defined: Upper and lower dimensions are output, as well as the determined deviation along the normal vector
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- In order to obtain exact results from the touch probe being used, you need to perform 3D calibration before executing Cycle **444**. Software option 92, **3D-ToolComp**, is required for 3D calibration.
- Cycle **444** generates a measuring log in HTML format.
- An error message is output if Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, or Cycle **26 AXIS-SPECIFIC SCALING** is active before Cycle **444** is run.
- For probing, an active TCPM will be taken into account. While the TCPM is active, probing of positions is possible even if the position resulting from the **Tilt working plane** function is inconsistent with the current position of the rotary axes.
- If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.
- Cycle **444** references all coordinates to the input system.
- The control writes the measured values to return parameters.

  Further information: "Application", Page 1837
- The workpiece status good/rework/scrap is set via Q parameter **Q183**, independent of parameter **Q309**.

  Further information: "Application", Page 1837

Note regarding machine parameters

- Depending on the setting of the optional machine parameter **chkTiltingAxes** (no. 204600), the control will check during probing whether the position of the rotary axes matches the tilting angles (3D-ROT). If that is not the case, the control displays an error message.

Cycle parameters

Help graphic	Parameter
	Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q581 Surface-normal in ref. axis? Enter here the surface normal in the direction of the main axis. The surface normal of a point is normally output by a CAD/CAM system. Input: **-10...+10**
	Q582 Surface-normal in minor axis? Enter here the surface normal in the direction of the secondary axis. The surface normal of a point is normally output by a CAD/CAM system. Input: **-10...+10**
	Q583 Surface-normal in tool axis? Enter here the surface normal in the direction of the tool axis. The surface normal of a point is normally output by a CAD/CAM system. Input: **-10...+10**
	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** or **PREDEF**
	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** or **PREDEF**

Help graphic	Parameter
	QS400 Tolerance value? Specify a tolerance band that will be monitored by the cycle. The tolerance defines the deviation permitted along the surface normal. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. (The surface normal is defined by **Q581** to **Q583**, and the nominal coordinate is defined by **Q263**, **Q264**, and **Q294**.) The tolerance value is distributed over the axes, depending on the normal vector (see examples). **Examples** ■ **QS400 ="0.4-0.1"** means: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate –0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate –0.1" ■ **QS400 ="0.4"** means: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate". ■ **QS400 ="-0.1"** means: Upper dimension = nominal coordinate; lower dimension = nominal coordinate –0.1. The following tolerance band thus results for the cycle: "nominal coordinate" to "nominal coordinate -0.1". ■ **QS400 =" "** means: No tolerance band. ■ **QS400 ="0"** means: No tolerance band. ■ **QS400 ="0.1+0.1"** means: No tolerance band. Input: Max. **255** characters
	Q309 Reaction to tolerance error? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run when tolerance is exceeded; do not output an error message **1**: Interrupt program run when tolerance is exceeded and output an error message **2:** If the value of the measured actual coordinate along the surface normal vector is less than the nominal coordinate, the control displays a message and interrupts the NC program run. However, there will be no error message if the value of the measured actual coordinate is greater than the nominal coordinate. Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 444 PROBING IN 3-D ~
    Q263=+0                ;1ST POINT 1ST AXIS ~
    Q264=+0                ;1ST POINT 2ND AXIS ~
    Q294=+0                ;1ST POINT 3RD AXIS ~
    Q581=+1                ;NORMAL IN REF. AXIS ~
    Q582=+0                ;NORMAL IN MINOR AXIS ~
    Q583=+0                ;NORMAL IN TOOL AXIS ~
    Q320=+0                ;SAFETY CLEARANCE ~
    Q260=+100              ;CLEARANCE HEIGHT ~
    QS400="1-1"            ;TOLERANCE ~
    Q309=+0                ;ERROR REACTION
```

31.5.5 Cycle 441 FAST PROBING

ISO programming
G441

Application

You can use touch probe cycle **441** to globally specify various touch probe parameters (e.g., the positioning feed rate) for all subsequently used touch probe cycles.

> The purpose of Cycle **441** is to set parameters for probing cycles. In this cycle, no machine movements will be performed.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- **END PGM**, **M2**, **M30** reset the global settings of Cycle **441**.
- Cycle parameter **Q399** depends on your machine configuration. Your machine manufacturer is responsible for the setting of whether the touch probe can be oriented through an NC program.
- Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397**=1.

Note regarding machine parameters

- The machine parameter **maxTouchFeed** (no. 122602) allows the machine manufacturer to limit the feed rate. You define the maximum absolute feed rate in this machine parameter.

Cycle parameters

Help graphic	Parameter
	Q396 Positioning feed rate? Define the feed rate at which the touch probe will be moved to the specified positions. Input: **0...99999.999**
	Q397 Pre-pos. at machine's rapid? Define whether the control, when prepositioning the touch probe, traverses at **FMAX** feed rate (machine's rapid traverse): **0**: Pre-position at the feed rate from **Q396** **1**: Pre-position at the machine's rapid traverse **FMAX** Input: **0**, **1**
	Q399 Angle tracking (0/1)? Define whether the control will orient the touch probe before every probing operation: **0**: Do not orient the spindle **1**: Orient the spindle before every probing operation (increased accuracy)? Input: **0**, **1**
	Q400 Automatic interruption? Define whether the control will interrupt program run and output the measurement results on the screen following a touch probe cycle for automatic workpiece measurement: **0**: Do not interrupt program run even if, in the specific touch probe cycle, the output of measurement results on the screen is selected **1**: Interrupt program run and output measurement results on the screen. You can then resume the NC program run with **NC Start**. Input: **0**, **1**

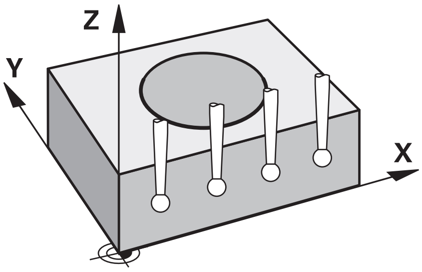
Example

```
11 TCH PROBE 441 FAST PROBING ~
    Q396=+3000              ;POSITIONING FEEDRATE ~
    Q397=+0                 ;SELECT FEED RATE ~
    Q399=+1                 ;ANGLE TRACKING ~
    Q400=+1                 ;INTERRUPTION
```

31.5.6 Cycle 1493 EXTRUSION PROBING

ISO programming
G1493

Application



Cycle **1493** allows you to repeat the touch points of specific touch probe cycles along a straight line. In the cycle, you define the direction and the length of the extrusion, as well as the number of extrusion points.

The repetitions allow you, for example, to perform multiple measurements at different heights and to determine deviations based on the deflection of the tool. You can also use the extrusion to increase the accuracy during probing. Multiple measuring points help you ascertain contamination on the workpiece or rough surfaces.

In order to activate the repetition of specific touch points, you need to define Cycle **1493** before the probing cycle. Depending on the definition, this cycle will remain active for only the next cycle or for the entire NC program. The control interprets the extrusion in the input coordinate system **I-CS**.

The following cycles are capable of performing extrusions:

- **PROBING IN PLANE** (Cycle **1420**, ISO: **G1420**, option 17), see Page 1612
- **PROBING ON EDGE** (Cycle **1410**, ISO: **G1410**), see Page 1618
- **PROBING TWO CIRCLES** (Cycle **1411**, ISO: **G1411**), see Page 1625
- **INCLINED EDGE PROBING** (Cycle **1412**, ISO: **G1412**), see Page 1633
- **INTERSECTION PROBING** (Cycle **1416**, ISO: **G1416**), see Page 1641
- **POSITION PROBING** (Cycle **1400**, ISO: **G1400**), see Page 1677
- **CIRCLE PROBING** (Cycle **1401**, ISO: **G1401**), see Page 1681
- **PROBE SLOT/RIDGE** (Cycle **1404**, ISO: **G1404**), see Page 1690
- **PROBE POSITION OF UNDERCUT** (Cycle **1430**, ISO: **G1430**), see Page 1695
- **PROBE SLOT/RIDGE UNDERCUT** (Cycle **1434**, DIN/ISO: **G1434**), see Page 1700

Result parameters

The control stores the results of the probing cycle in the following Q parameters:

Q parameter number	Meaning
Q970	Maximum deviation from the ideal line of touch point 1
Q971	Maximum deviation from the ideal line of touch point 2
Q972	Maximum deviation from the ideal line of touch point 3
Q973	Maximum deviation of diameter 1
Q974	Maximum deviation of diameter 2

QS parameter

In addition to saving the results of the probing cycle in the return parameters **Q97x**, the control saves individual results to the QS parameters **QS97x**. The control saves the results of all measuring points from **one** extrusion in the corresponding QS parameters. Each result is ten characters long and the results are separated from each other by a blank space. This makes it easy for the control to convert the individual values in the NC program via string processing and use them for special automated evaluations.

Result in a QS parameter:

QS970 = "0.12345678 -1.1234567 -2.1234567 -3.12345678"

Further information: "String functions", Page 1400

Log function

Once probing has finished, the control generates a log file in HTML format. The log file contains the results of the 3D deviation in graphical and tabular form. The control saves the log file in the same folder in which the NC program is located.

The log file contains the following data in the main axis, secondary axis and tool axis depending on the selected cycle (e.g., circle center point and diameter):

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- Upper and lower dimensions, as well as the determined deviation along the normal vector
- Measured actual coordinate
- Color coding of the values:
  - Green: Good
  - Orange: Rework
  - Red: Scrap
- Extrusion points

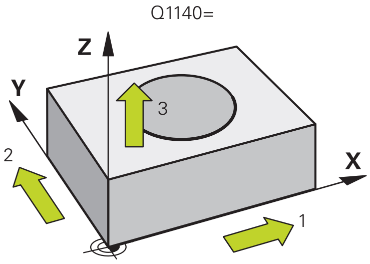
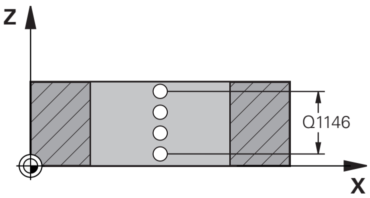
Extrusion points:

The horizontal axis represents the direction for the extrusion. The blue points are the individual measuring points. The red lines indicate the lower limit and the upper limit of the dimensions. If a value violates a specified tolerance, the control will show the area in red color in the graphic.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If **Q1145 > 0** and **Q1146 = 0**, then the control will perform the number of extrusion points at the same position.
- If you use Cycle **1401 CIRCLE PROBING** or **1411 PROBING TWO CIRCLES** to perform an extrusion, the direction for the extrusion must be **Q1140 = +3**; otherwise, the control will output an error message.

Cycle parameters

Help graphic	Parameter
	**Q1140 Direction for extrusion (1-3)?** **1:** Extrusion in the direction of the main axis **2:** Extrusion in the direction of the secondary axis **3:** Extrusion in the direction of the tool axis Input: **1**, **2**, **3**
	Q1145 Number of extrusion points? Number of measuring points that the cycle repeats over the length of the extrusion **Q1146**. Input: **1...99**
	**Q1146 Length of extrusion?** Length over which the measuring points are repeated. Input: **-99...+99**
	Q1149 Extrusion: Modal duration? Effect of the cycle: **0**: The extrusion is effective for only the next cycle. **1:** The extrusion is effective until the end of the NC program. Input: **-99...+99**

Example

```
11 TCH PROBE 1493 EXTRUSION PROBING ~
    Q1140=+3            ;EXTRUSION DIRECTION ~
    Q1145=+1            ;EXTRUSION POINTS ~
    Q1146=+0            ;EXTRUSION LENGTH ~
    Q1149=+0            ;EXTRUSION MODAL
```

31.6 Touch Probe Cycles: Calibration

31.6.1 Fundamentals

Overview

> 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.

In order to precisely specify the actual trigger point of a 3D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.

> Always calibrate a touch probe in the following cases:
> - Initial configuration
> - Broken stylus
> - Stylus replacement
> - Change in the probe feed rate
> - Irregularities caused, for example, when the machine heats up
> - Change of active tool axis
>
> The control assumes the calibration values for the active touch probe directly after the calibration process. The updated tool data are immediately effective. It is not necessary to repeat the tool call.

During calibration, the control determines the effective length of the stylus and the effective radius of the stylus tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.
The control provides calibration cycles for calibrating the length and the radius:

Cycle		Call	Further information
461	**TS CALIBRATION OF TOOL LENGTH** ■ Calibrating the length	DEF-active	Page 1850
462	**CALIBRATION OF A TS IN A RING** ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge	DEF-active	Page 1851
463	**TS CALIBRATION ON STUD** ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin	DEF-active	Page 1855
460	**CALIBRATION OF TS ON A SPHERE** ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere	DEF-active	Page 1858

Calibrating a touch trigger probe

In order to precisely specify the actual trigger point of a 3D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.

Always calibrate a touch probe in the following cases:

- Initial configuration
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The control provides calibration cycles for calibrating the length and the radius.

> - The control applies the calibration values for the active probe system directly after the calibration process. The updated tool data are immediately effective. It is not necessary to repeat the tool call.
> - Ensure that the touch probe number in the tool table and the touch-probe number in the touch-probe table are identical.
>
> **Further information:** "Touch probe table tchprobe.tp", Page 2022

Displaying calibration values

The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (main axis) and **CAL_OF2** (secondary axis).

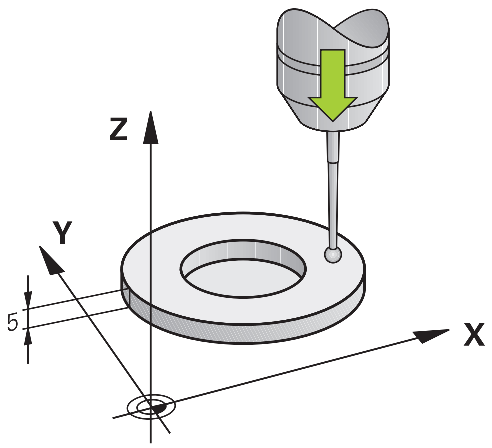
A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

31.6.2 Cycle 461 TS CALIBRATION OF TOOL LENGTH

ISO programming
G461

Application

> Refer to your machine manual.



Before starting the calibration cycle, you must set the preset in the spindle axis so that Z=0 on the machine table; you must also pre-position the touch probe above the calibration ring.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

Cycle sequence

1 The control orients the touch probe to the angle **CAL_ANG** specified in the touch probe table (only if your touch probe can be oriented).
2 The control probes from the current position in the negative spindle axis direction at the probing feed rate (column **F** from the touch probe table).
3 The control then retracts the touch probe at rapid traverse (column **FMAX** from the touch probe table) to the starting position.

Notes

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

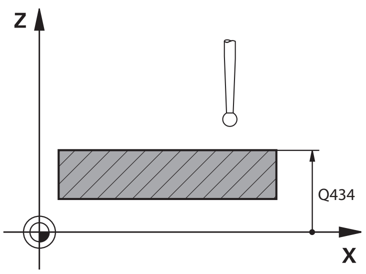
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Cycle parameters

Help graphic	Parameter
	**Q434 Preset for length?** Preset for the length (e.g., height of the calibration ring). This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH ~
    Q434=+5                     ;PRESET
```

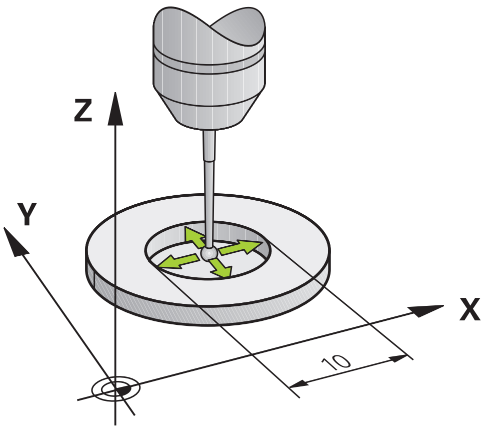
31.6.3 Cycle 462 CALIBRATION OF A TS IN A RING

ISO programming

G462

Application

⚙	Refer to your machine manual.



Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows a reversal measurement, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The reversal measurement determines not only the radius but ralso the center offset (**CAL_OF** in the touch-probe table).
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Notes

In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

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!

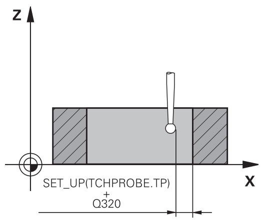
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q407 Radius of ring gauge?** Enter the radius of the ring gauge. Input: **0.0001...99.9999**
	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** or **PREDEF**
	Q423 Number of probes? Number of measuring points on the diameter. This value has an absolute effect. Input: **3...8**
	Q380 Ref. angle in ref. axis? Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **0...360**

Example

```
11 TCH PROBE 462 CALIBRATION OF A TS IN A RING ~
    Q407=+5                 ;RING RADIUS ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q423=+8                 ;NO. OF PROBE POINTS ~
    Q380=+0                 ;REFERENCE ANGLE
```

31.6.4 Cycle 463 TS CALIBRATION ON STUD

ISO programming
G463

Application

⚙	Refer to your machine manual.

Before starting the calibration cycle, you need to pre-position the touch probe above the center of the calibration pin. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration pin.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or stud (approximate measurement) and positions the touch probe in the center. Then, during the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe permits a reversal measurement, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column **R** in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The reversal measurement determines now only the radius but also the center offset (CAL_OF in the touch-probe table).
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions"

Note:

In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

Whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

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!

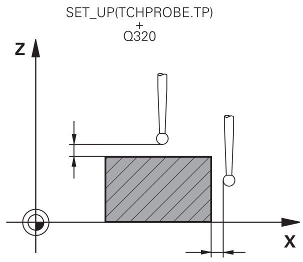
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q407 Radius of calibr. stud?** Diameter of the calibration stud Input: **0.0001...99.9999**
	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** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q423 Number of probes? Number of measuring points on the diameter. This value has an absolute effect. Input: **3...8**
	Q380 Ref. angle in ref. axis? Angle between the main axis of the working plane and the first touch point. This value has an absolute effect. Input: **0...360**

Example

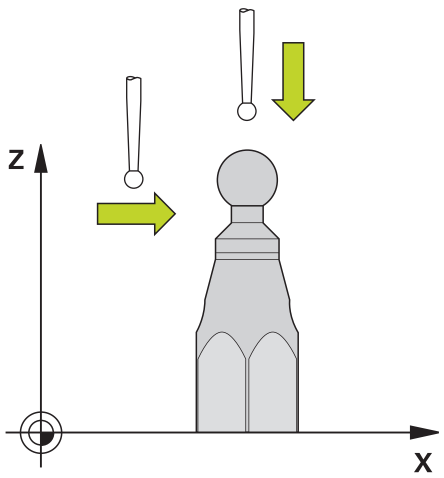
```
11 TCH PROBE 463 TS CALIBRATION ON STUD ~
    Q407=+5              ;STUD RADIUS ~
    Q320=+0              ;SET-UP CLEARANCE ~
    Q301=+1              ;MOVE TO CLEARANCE ~
    Q423=+8              ;NO. OF PROBE POINTS ~
    Q380=+0              ;REFERENCE ANGLE
```

31.6.5 Cycle 460 CALIBRATION OF TS ON A SPHERE (option 17)

ISO programming
G460

Application

⚙	Refer to your machine manual.



Before starting the calibration cycle, you need to pre-position the touch probe above the center of the calibration sphere. Position the touch probe in the touch probe axis by approximately the amount of the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere.

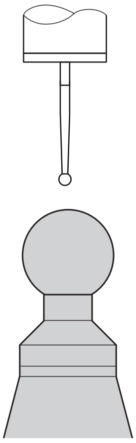
With Cycle **460** you can calibrate a triggering 3D touch probe automatically using an exact calibration sphere.

It is also possible to capture 3D calibration data. Option 92, 3D-ToolComp, is required for this purpose. 3D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3D calibration data are stored under TNC:\system\3D-ToolComp*. The **DR2TABLE** column of the tool table references the 3DTC table. The 3D calibration data are then taken into account when probing. This 3D calibration is necessary if you want to achieve very high accuracy, for example with Cycle **444** or if you want to graphically align the workpiece (option 159).

Before calibrating with a normal stylus:

Before starting the calibration cycle, you need to pre-position the touch probe:

- Define the approximate value of the radius R and length L of the touch probe
- In the working plane, center the touch probe above the calibration sphere
- Position the touch probe in the touch probe axis by approximately the amount of the set-up clearance above the calibration sphere. The set-up clearance consists of the value from the touch probe table plus the value from the cycle.



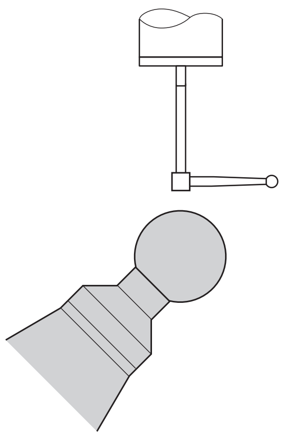
Pre-positioning with a normal stylus

Before calibrating with an L-shaped stylus:

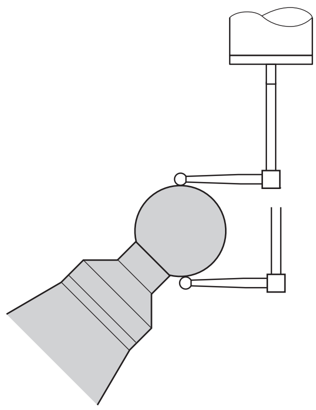
- Clamp the calibration sphere

> It must be possible to probe the north pole and south pole during calibration. If this is not possible, the control cannot determine the sphere radius. Ensure that no collision can occur.

- Define the approximate value of the radius **R** and length **L** of the touch probe. You can determine these with a tool presetter.
- Enter the approximate center offset in the touch probe table:
  - **CAL_OF1**: length of the extension
  - **CAL_OF2**: **0**
- Insert the touch probe and orient it parallel to the main axis, for example with Cycle **13 ORIENTATION**
- Enter the calibration angle in the **CAL_ANG** column of the tool table.
- Position the center of the touch probe over the center of the calibration sphere
- > Since the stylus is angled, the touch probe sphere is not centered over the calibration sphere.
- Position the touch probe in the tool axis by approximately the amount of the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere

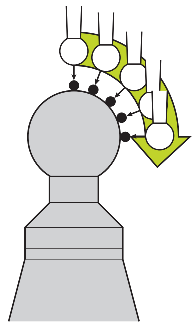


Pre-positioning with an L-shaped stylus



Calibration process with an L-shaped stylus

Cycle sequence



The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

Radius calibration Q433=0

1. Clamp the calibration sphere. Ensure the prevention of collisions
2. In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center
3. The first movement is in the plane, depending on the reference angle (**Q380**)
4. The controls positions the touch probe in the touch probe axis
5. The probing process starts, and the control begins by searching for the equator of the calibration sphere
6. Once the equator has been determined, the determination of the spindle angle for calibration **CAL_ANG** begins (for L-shaped stylus)
7. Once **CAL_ANG** has been determined, the radius calibration begins
8. Finally, the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned

Radius and length calibration Q433=1

1. Clamp the calibration sphere. Ensure the prevention of collisions
2. In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center
3. The first movement is in the plane, depending on the reference angle (**Q380**)
4. The control then positions the touch probe in touch-probe axis
5. The probing process starts, and the control begins by searching for the equator of the calibration sphere
6. Once the equator has been determined, the determination of the spindle angle for calibration **CAL_ANG** begins (for L-shaped stylus)
7. Once **CAL_ANG** has been determined, the radius calibration begins
8. The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned
9. The control determines the length of the touch probe at the north pole of the calibration sphere
10. At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned

The setting in parameter **Q455** specifies whether you can perform an additional 3D calibration

3D calibration Q455= 1...30

1. Clamp the calibration sphere. Ensure the prevention of collisions
2. After calibration of the radius and length, the control retracts the touch probe in touch-probe axis. Then the control positions the touch probe above the north pole
3. The probing process goes from the north pole to the equator in several steps. Deviations from the nominal value, and therefore the specific deflection behavior, are determined
4. You can specify the number of touch points between the north pole and the equator. This number depends on input parameter **Q455**. A value between 1 and 30 can be programmed. If you program **Q455**=0, no 3D calibration will be performed
5. The deviations determined during the calibration are stored in a 3DTC table
6. At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned

- For an L-shaped stylus, the calibration takes place between the north and south pole.
- In order to calibrate the length, the position of the center point (**Q434**) of the calibration sphere relative to the active datum must be known If this is not the case, then performing length calibration with Cycle **460** is not recommended!
- One application example for calibrating the length with Cycle **460** is the comparison of two touch probes

Notes

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

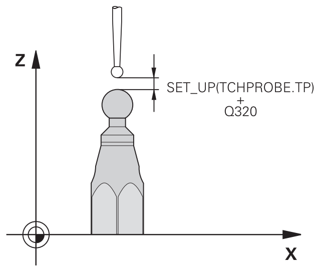
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- Depending on the accuracy of the pre-positioning, finding the equator of the calibration sphere will require a different number of touch points.
- In order to achieve optimum accuracy results with an L-shaped stylus, HEIDENHAIN recommends calibrating and probing at identical speeds. Note the setting of the feed override if it is active for probing.
- If you program **Q455=0**, the control will not perform a 3D calibration.
- If you program **Q455=1** to **30**, the control will perform a 3D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles. If you use Cycle **444**, you should first perform a 3D calibration.
- If you program **Q455=1** to **30**, a table will be stored under TNC:\system\3D-ToolComp*.
- If there is already a reference to a calibration table (entry in **DR2TABLE**), this table will be overwritten.
- If there is no reference to a calibration table (entry in **DR2TABLE**), then, in dependence of the tool number, a reference and the associated table will be created.

Note on programming

- Before a cycle definition you must program a tool call to define the touch-probe axis.

Cycle parameters

Cycle parameters

Help graphic	Parameter
	**Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: **0:** Move at measuring height between measuring points **1:** Move at clearance height between measuring points Input: **0**, **1**
	Q423 Number of probes? Number of measuring points on the diameter. This value has an absolute effect. Input: **3...8**
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: **0...360**
	Q433 Calibrate length (0/1)? Define whether the control will calibrate the touch probe length after radius calibration: **0**: Do not calibrate touch probe length **1**: Calibrate touch probe length Input: **0**, **1**
	Q434 Preset for length? Coordinate of the calibration sphere center. This value must be defined only if length calibration will be carried out. This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q455 No. of points for 3-D calibrtn.? Enter the number of touch points for 3D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3D calibration. During 3D calibration, the deflecting behavior of the touch probe is determined under various angles, and the values are stored in a table. 3D-ToolComp is required for 3D calibration. Input: **0...30**

Example

```
11 TCH PROBE 460 TS CALIBRATION OF TS ON A SPHERE ~
    Q407=+12.5              ;SPHERE RADIUS ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q301=+1                 ;MOVE TO CLEARANCE ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q380=+0                 ;REFERENCE ANGLE ~
    Q433=+0                 ;CALIBRATE LENGTH ~
    Q434=-2.5               ;PRESET ~
    Q455=+15                ;NO. POINTS 3-D CAL.
```

31.7 Touch Probe Cycles: Automatic Kinematics Measurement

31.7.1 Fundamentals (option 48)

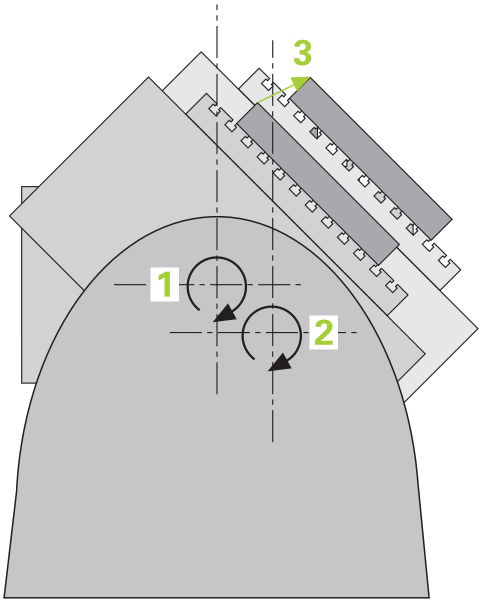
Overview

> 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 control offers the following cycles that enable you to automatically save, restore, check, and optimize the machine kinematics:

Cycle		Call	Further information
450	**SAVE KINEMATICS** (option 48) ■ Storing the active machine kinematic configuration ■ Restoring previously saved kinematic configuration	**DEF**-active	Page 1870
451	**MEASURE KINEMATICS** (option 48) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the machine kinematic configuration	**DEF**-active	Page 1873
452	**PRESET COMPENSATION** (option 48) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the kinematic transformation chain of the machine	**DEF**-active	Page 1888
453	**KINEMATICS GRID** (option 48, option 52) ■ Automatic checking depending on the rotary axis position of the machine kinematic configuration ■ Optimizing the machine kinematic configuration	**DEF**-active	Page 1899

Fundamentals



Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure) and the kinematic conditions actually existing on the machine (see **2** in the figure). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure). It is therefore necessary for the model to approach reality as closely as possible.

The **KinematicsOpt** function of the control is an important component that helps you meet these complex requirements in real life: a 3D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, a calibration sphere is attached at any position on the machine table, and measured with a resolution that you define. During cycle definition, you simply define for each rotary axis the area that you want to measure.

From the measured values, the control calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics table.

Requirements

Refer to your machine manual.

Advanced Function Set 1 (option 8) must be enabled.

Option 48 must be enabled.

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

Requirements for using KinematicsOpt:

The machine manufacturer must have defined the machine parameters for **CfgKinematicsOpt** (no. 204800) in the configuration data.

- **maxModification** (no. 204801) specifies the tolerance limit starting from which the control is to display a message if the changes made to the kinematic data exceed this limit value
- **maxDevCalBall** (no. 204802) defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter
- **mStrobeRotAxPos** (no. 204803) defines an M function that is specifically configured by the machine manufacturer and is used to position the rotary axes

- The 3D touch probe used for the measurement must be calibrated
- The cycles can only be carried out with the tool axis Z
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table
- The kinematics description of the machine must be complete and correct, and the transformation dimensions must have been entered with an accuracy of approx. 1 mm
- The complete machine geometry must have been measured (by the machine manufacturer during commissioning)

HEIDENHAIN recommends using the calibration spheres **KKH 250 (ordering number: 655475-01)** or **KKH 80 (ordering number: 655475-03)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

Notes

HEIDENHAIN only guarantees the proper operation of the probing cycles if HEIDENHAIN touch probes are used.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

A change in the kinematics always changes the preset as well. Basic rotations will automatically be reset to 0. There is a danger of collision!

- After an optimization, reset the preset

Notes about machine parameters

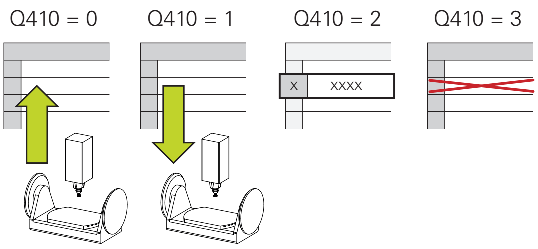
- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the position of the rotary axes. If an M function has been defined in the machine parameter, you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for **450**).
- If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

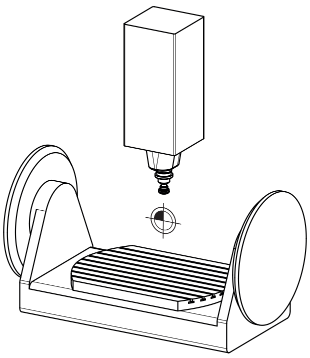
31.7.2 Cycle 450 SAVE KINEMATICS (option 48)

ISO programming
G450

Application

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





With touch probe cycle **450** you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.

Notes

> Only save and restore data with Cycle **450** while no tool carrier kinematics configuration that includes transformations is active.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Always save the active kinematic model before running a kinematics optimization.

  Advantage:
  - You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g., power failure).
- With the **Restore** mode, note the following:
  - The control can restore saved data only to a matching kinematic configuration
  - A change in the kinematics always changes the preset as well. So redefine the preset, if required.
- The cycle does not restore identical values. It only restores values that differ from the present values. Compensations can only be restored if they had been saved before.

Notes on data management

The control stores the saved data in the file **TNC:\table\DATA450.KD**. This file can be backed up to an external PC with **TNCremo**, for example. If you delete the file, the stored data are removed, too. If the data in the file are changed manually, the data records may become corrupted so that they are unusable.

> Operating notes:
> - If the file **TNC:\table\DATA450.KD** does not exist, it is generated automatically when Cycle **450** is run.
> - Make sure that you delete any empty files with the name **TNC:\table\DATA450.KD** before starting Cycle **450**. If there is an empty memory table (**TNC:\table\DATA450.KD**) without any rows in it, an error message will be issued when running Cycle **450**. In this case, delete the empty memory table and call the cycle again.
> - Do not change stored data manually.
> - Make a backup of the **TNC:\table\DATA450.KD** file so that you can restore the file, if necessary (e.g. if the data medium is damaged).

Cycle parameters

Help graphic	Parameter
	Q410 Mode (0/1/2/3)? Define whether a kinematic model will be saved or restored: **0**: Save active kinematics **1**: Restore saved kinematics **2**: Display the current memory status **3**: Delete a data record Input: **0**, **1**, **2**, **3**
	Q409/QS409 Name of data record? Number or name of data record identifier. **Q409** does not function if mode 2 has been selected. Wildcards can be used for searches in modes 1 and 3 (Restore and Delete). If the control finds several possible data records because of the wildcards, the control restores the mean values of the data (mode 1) or deletes all selected data records after confirmation (mode 3). You can use the following wildcards in searches: **?**: A single, undefined character **$**: A single alphabetic character (letter) **#**: A single, undefined number *****: An undefined string of any length Input: **0...99999** or max. **255** characters. A total of 16 memory locations are available.

Saving the current kinematics

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+0                  ;MODE ~
    Q409=+947                ;MEMORY DESIGNATION
```

Restoring data records

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+1                  ;MODE ~
    Q409=+948                ;MEMORY DESIGNATION
```

Displaying all saved data records

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+2                  ;MODE ~
    Q409=+949                ;MEMORY DESIGNATION
```

Deleting data records

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+3                  ;MODE ~
    Q409=+950                ;MEMORY DESIGNATION
```

Log function

After running Cycle **450**, the control creates a log (**TCHPRAUTO.html**) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

The other data in the log vary depending on the selected mode:

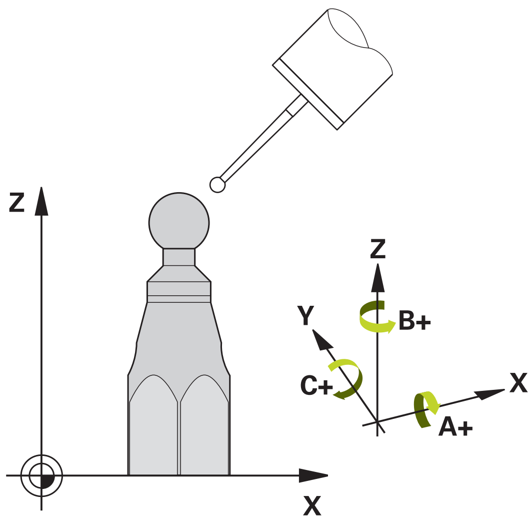
- Mode 0: Logging of all axis entries and transformation entries of the kinematics chain that the control has saved.
- Mode 1: Logging of all transformation entries before and after restoring the kinematics configuration.
- Mode 2: List of the saved data records
- Mode 3: List of the deleted data records

31.7.3 Cycle 451 MEASURE KINEMATICS (option 48)

ISO programming
G451

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



Touch probe cycle **451** enables you to check and, if required, optimize the kinematics of your machine. Use the 3D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

The control will determine the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

Cycle sequence

1. Clamp the calibration sphere and check for potential collisions.
2. In **Manual operation** mode, set the preset to the center of the sphere or, if you defined **Q431** = 1 or **Q431** = 3: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3. Select the Program Run operating mode and start the calibration program.
4. The control automatically measures all rotary axes successively in the resolution you defined.

Programming and operating notes:

- If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.
- During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

The control saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (–1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (–1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. A reference measurement is automatically performed at 0°.

Specify the start and end angles in such a way that the same position is not measured twice. A duplicated point measurement (e.g., measuring positions +90° and -270°) is not advisable, but it will not generate an error message.

- Example: Start angle = +90°, end angle = –90°
  - Start angle = +90°
  - End angle = –90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (-90° - +90°) / (4 – 1) = -60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +30°
  - Measuring point 3 = -30°
  - Measuring point 4 = -90°
- Example: start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (270° – 90°) / (4 – 1) = +60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +150°
  - Measuring point 3 = +210°
  - Measuring point 4 = +270°

Machines with Hirth-coupled axes

NOTICE

Danger of collision!

In order to be positioned, the axis must move out of the Hirth grid. If necessary, the control rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points). There is a danger of collision!

- So remember to leave a large enough set-up clearance to prevent any risk of collision between the touch probe and calibration sphere
- Also ensure that there is enough space to reach the set-up clearance (software limit switch)

NOTICE

Danger of collision!

Depending on the machine configuration, the control cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine manufacturer, enabling the control to move the rotary axes. The machine manufacturer must have entered the number of the M function in the machine parameter **mStrobeRotAxPos** (no. 204803) for this purpose. There is a danger of collision!

- Note the documentation of the machine manufacturer

- Define a retraction height greater than 0 if option 2 is not available.
- The measured positions are calculated from the start angle, end angle, and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = –30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = (**Q412** – **Q411**) / (**Q414** – 1)

Calculated stepping angle = (90° - (-30°)) / (4 – 1) = 120 / 3 = 40°

Measuring position 1 = **Q411** + 0 * stepping angle = –30° --> –30°

Measuring position 2 = **Q411** + 1 * stepping angle = +10° --> 9°

Measuring position 3 = **Q411** + 2 * stepping angle = +50° --> 51°

Measuring position 4 = **Q411** + 3 * stepping angle = +90° --> 90°

Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1 or 2), for example when commissioning the machine.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°. Thus, define a starting angle of 90° and an end angle of 270°.

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the **Check** mode.

> If a measuring point has been defined at 0°, it will be ignored because the reference measurement is always done at 0°.

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machines with rotary tables/tilting tables: Clamp the calibration sphere as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
  - Position the calibration sphere in such a way on the machine table that, with rotary table axes, there is a large measuring circle or that, on swivel head axes, measurement can be made at a representative position (e.g., in the center of the traverse range).
- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made with the aid of the inclination angle of an axis (**Q413**/**Q417**/**Q421**) around the rotary axis angle at which the workpiece is to be machined later.
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.
- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes.

Notes on the accuracy

If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine manual provides further information.

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion output by the control in the measurement log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.

If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is performed with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432**, you can activate backlash measurement. Enter an angle that the control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.

Backlash measurement is not possible if an M function for positioning the rotary axes is set in the optional **mStrobeRotAxPos** machine parameter (no. 204803) or if the axis is a Hirth axis.

Programming and operating notes:

- The control does not perform an automatic backlash compensation.
- If the measuring circle radius is < 1 mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can ascertain the rotary axis backlash.

  Further information: "Log function", Page 1887

Notes

Angle compensation is only possible with option 52 KinematicsComp.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- Deactivate the basic rotation before running the cycle.
- Set the preset and the basic rotation again after optimization.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- The control ignores cycle definition data that applies to inactive axes.
- A correction in the machine datum (**Q406**=3) is only possible if superimposed rotary axes on the spindle head side or table side are measured.
- If you have activated presetting before the calibration (**Q431** = 1/3), then move the touch probe to the set-up clearance (**Q320** + SET_UP) to a position approximately above the center of the calibration sphere before the start of the cycle.
- Programming in inches: The control always records the log data and results of measurement in millimeters.
- After measuring the kinematics, you must re-determine the preset.

Notes about machine parameters

- If the optional machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to –1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.
- For angle optimization, the machine manufacturer must adapt the configuration correspondingly.

Cycle parameters

Help graphic	Parameter
	Q406 Mode (0/1/2/3)? Define whether the control will check or optimize the active kinematics: **0**: Check the active machine kinematics. The control measures the kinematics in the rotary axes you have defined, but it does not make any changes to the active kinematics. The control displays the measurement results in a measuring log. **1**: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **the rotary axes positions** of the active kinematics. **2**: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **angle and position errors**. Software option 52, KinematicsComp, is required for compensation of angle errors. **3**: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then automatically compensates the machine datum. It then optimizes **angle and position errors**. Software option 52, KinematicsComp, is required. Input: **0**, **1**, **2**, **3**
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	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** or **PREDEF**
	Q408 Retraction height? **0**: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. **> 0**: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**. This value has an absolute effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: **0...360**
	Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**
	Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: **-359.999...+360.000**

Help graphic	Parameter
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision. Input: **3...8**
	Q431 Preset (0/1/2/3)? Define whether the control will automatically set the active preset at the center of the sphere: **0**: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle **1**: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle **2**: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle **3**: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	Q432 Angular range of backlash comp.? Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input: **-3...+3**

Saving and checking the kinematics

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+0                ;MODE ~
    Q409=+5                ;MEMORY DESIGNATION
13 TCH PROBE 451 MEASURE KINEMATICS ~
    Q406=+0                ;MODE ~
    Q407=+12.5             ;SPHERE RADIUS ~
    Q320=+0                ;SET-UP CLEARANCE ~
    Q408=+0                ;RETR. HEIGHT ~
    Q253=+750              ;F PRE-POSITIONING ~
    Q380=+0                ;REFERENCE ANGLE ~
    Q411=-90               ;START ANGLE A AXIS ~
    Q412=+90               ;ENDWINKEL A-ACHSE ~
    Q413=+0                ;INCID. ANGLE A AXIS ~
    Q414=+0                ;MEAS. POINTS A AXIS ~
    Q415=-90               ;START ANGLE B AXIS ~
    Q416=+90               ;END ANGLE B AXIS ~
    Q417=+0                ;INCID. ANGLE B AXIS ~
    Q418=+2                ;MEAS. POINTS B AXIS ~
    Q419=-90               ;START ANGLE C AXIS ~
    Q420=+90               ;END ANGLE C AXIS ~
    Q421=+0                ;INCID. ANGLE C AXIS ~
    Q422=+2                ;MEAS. POINTS C AXIS ~
    Q423=+4                ;NO. OF PROBE POINTS ~
    Q431=+0                ;PRESET ~
    Q432=+0                ;BACKLASH, ANG. RANGE
```

Various modes (Q406)

Test mode Q406 = 0

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control records the results of a possible position optimization but does not make any adjustments.

"Optimize position of rotary axes" mode Q406 = 1

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the control tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data are adjusted automatically.

Position and Angle Optimization mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- First the control tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

> Depending on the machine kinematics for correctly determining the angles, HEIDENHAIN recommends performing the measurement once with an inclination angle of 0°.

"Optimize machine datum, position, and angle" mode (Q406 = 3)

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control automatically tries to optimize the datum (KinematicsComp, option 52). In order to use a machine datum to compensate for the angular position of a rotary axis, the rotary axis to be compensated must be nearer to the machine base in the machine kinematics than the measured rotary axis.
- The control then tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

> - For correct determination of the angular position errors, HEIDENHAIN recommends setting the affected rotary axis to an inclination angle of 0° for this measurement.
> - After correcting a machine datum, the control tries to reduce the compensation of the associated angular position error (**locErrA**/**locErrB**/**locErrC**) of the measured rotary axis.

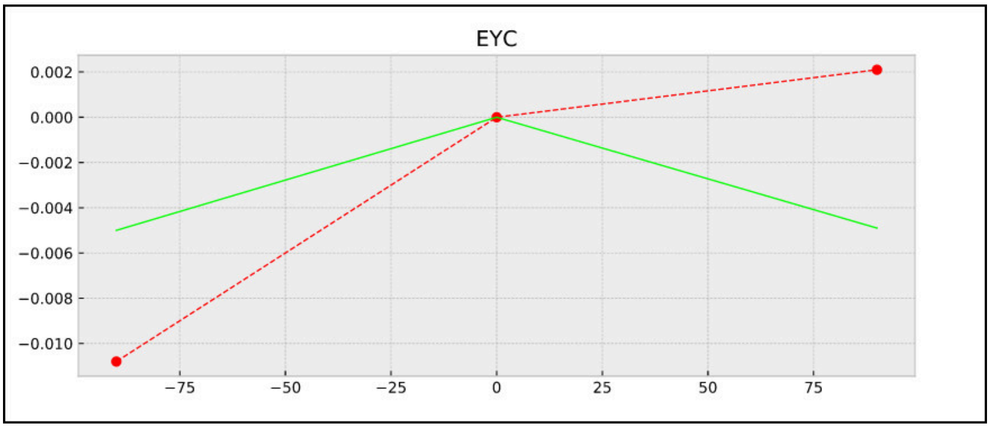
Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 451 MEASURE KINEMATICS ~
    Q406=+1              ;MODE ~
    Q407=+12.5           ;SPHERE RADIUS ~
    Q320=+0              ;SET-UP CLEARANCE ~
    Q408=+0              ;RETR. HEIGHT ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q380=+0              ;REFERENCE ANGLE ~
    Q411=-90             ;START ANGLE A AXIS ~
    Q412=+90             ;END ANGLE A AXIS ~
    Q413=+0              ;INCID. ANGLE A AXIS ~
    Q414=+0              ;MEAS. POINTS A AXIS ~
    Q415=-90             ;START ANGLE B AXIS ~
    Q416=+90             ;END ANGLE B AXIS ~
    Q417=+0              ;INCID. ANGLE B AXIS ~
    Q418=+4              ;MEAS. POINTS B AXIS ~
    Q419=+90             ;START ANGLE C AXIS ~
    Q420=+270            ;END ANGLE C AXIS ~
    Q421=+0              ;INCID. ANGLE C AXIS ~
    Q422=+3              ;MEAS. POINTS C AXIS ~
    Q423=+3              ;NO. OF PROBE POINTS ~
    Q431=+1              ;PRESET ~
    Q432=+0.5            ;BACKLASH, ANG. RANGE
```

Log function

After running Cycle 451, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Tool name
- Active kinematics
- Mode used (0=Check/1=Optimize position/2=Optimize pose/3=Optimize machine datum and pose)
- Inclination angles
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Number of measuring points
  - Measuring circle radius
  - Averaged backlash, if **Q423>0**
  - Positions of the axes
  - Angular position errors (only with **KinematicsComp**, option 52)
  - Standard deviation (scatter)
  - Maximum deviation
  - Angular error
  - Compensation values in all axes (preset shift)
  - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Averaged positioning error and standard deviation of the positioning errors to 0
  - SVG files with graphs: measured and optimized errors of individual measurement positions.
    - Red curve: measured positions
    - Green curve: optimized values after cycle has run
    - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = component error in Y of axis C)
    - X axis of the graph: rotary axis position in degrees
    - Y axis of the graph: position deviations in mm



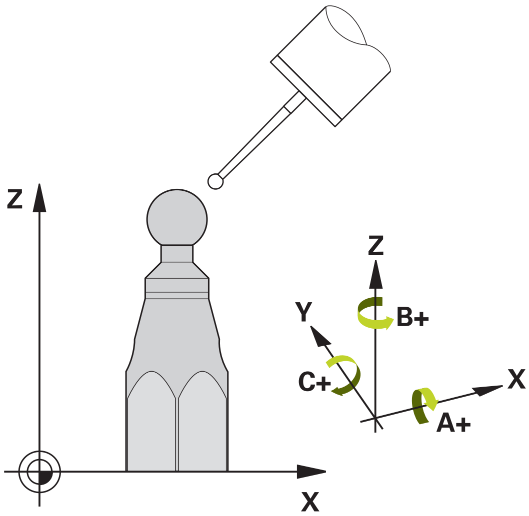
Sample measurement: EYC component error in Y of axis C

31.7.4 Cycle 452 PRESET COMPENSATION (option 48)

ISO programming
G452

Application

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



Touch probe cycle **452** optimizes the kinematic transformation chain of your machine (see "Cycle 451 MEASURE KINEMATICS (option 48)", Page 1873). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is at the center of the calibration sphere after optimization.

Cycle sequence

> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

1 Clamp the calibration sphere
2 Measure the complete reference head with Cycle **451**, and then use Cycle **451** to set the preset in the center of the sphere.
3 Insert the second head
4 Use Cycle **452** to measure the interchangeable head up to the point where the head is changed.
5 Use Cycle **452** to adjust other interchangeable heads to the reference head

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

1 Clamp the calibration sphere and check for potential collisions.
2 Set the preset in the calibration sphere.
3 Set the preset on the workpiece, and start machining the workpiece.
4 Use Cycle **452** for preset compensation at regular intervals. The control measures the drift of the axes involved and compensates it in the kinematics description.

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (–1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (–1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Notes

In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- Deactivate the basic rotation before running the cycle.
- Set the preset and the basic rotation again after optimization.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Ensure that all functions for tilting the working plane are reset.
- Before defining the cycle, you must set the preset at the center of the calibration sphere and activate it.
- For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse an angle of 1° to the limit switch. The control needs this traverse for internal backlash compensation.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while touch probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.

- If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle **450**, so that in case of a failure the most recently active kinematic configuration can be restored.

Notes about machine parameters

- In the machine parameter **maxModification** (no. 204801), the machine manufacturer defines the permissible limit value for modifications of a transformation. If the kinematics data determined exceed the permissible limit value, the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC Start**.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	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** or **PREDEF**
	Q408 Retraction height? **0**: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. **> 0**: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**. This value has an absolute effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: **0...360**
	Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**

Help graphic	Parameter
	Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: **-359.999...+360.000**
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. This value has an absolute effect. Input: **-359.9999...+359.9999**
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision. Input: **3...8**

Help graphic	Parameter
	Q432 Angular range of backlash comp.? Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input: **-3...+3**

Calibration program

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+0                   ;MODE ~
    Q409=+5                   ;MEMORY DESIGNATION
13 TCH PROBE 452 PRESET COMPENSATION ~
    Q407=+12.5                ;SPHERE RADIUS ~
    Q320=+0                   ;SET-UP CLEARANCE ~
    Q408=+0                   ;RETR. HEIGHT ~
    Q253=+750                 ;F PRE-POSITIONING ~
    Q380=+0                   ;REFERENCE ANGLE ~
    Q411=-90                  ;START ANGLE A AXIS ~
    Q412=+90                  ;END ANGLE A AXIS ~
    Q413=+0                   ;INCID. ANGLE A AXIS ~
    Q414=+0                   ;MEAS. POINTS A AXIS ~
    Q415=-90                  ;START ANGLE B AXIS ~
    Q416=+90                  ;END ANGLE B AXIS ~
    Q417=+0                   ;INCID. ANGLE B AXIS ~
    Q418=+2                   ;MEAS. POINTS B AXIS ~
    Q419=-90                  ;START ANGLE C AXIS ~
    Q420=+90                  ;END ANGLE C AXIS ~
    Q421=+0                   ;INCID. ANGLE C AXIS ~
    Q422=+2                   ;MEAS. POINTS C AXIS ~
    Q423=+4                   ;NO. OF PROBE POINTS ~
    Q432=+0                   ;BACKLASH, ANG. RANGE
```

Adjustment of interchangeable heads

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

- Load the second interchangeable head.
- Insert the touch probe
- Measure the interchangeable head with Cycle **452**
- Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with **Q422**)
- The preset and the position of the calibration sphere must not be changed during the entire process.
- All other interchangeable heads can be adjusted in the same way

Adjusting an interchangeable head

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 452 PRESET COMPENSATION ~
    Q407=+12.5              ;SPHERE RADIUS ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q408=+0                 ;RETR. HEIGHT ~
    Q253=+2000              ;F PRE-POSITIONING ~
    Q380=+45                ;REFERENCE ANGLE ~
    Q411=-90                ;START ANGLE A AXIS ~
    Q412=+90                ;END ANGLE A AXIS ~
    Q413=+45                ;INCID. ANGLE A AXIS ~
    Q414=+4                 ;MEAS. POINTS A AXIS ~
    Q415=-90                ;START ANGLE B AXIS ~
    Q416=+90                ;END ANGLE B AXIS ~
    Q417=+0                 ;INCID. ANGLE B AXIS ~
    Q418=+2                 ;MEAS. POINTS B AXIS ~
    Q419=+90                ;START ANGLE C AXIS ~
    Q420=+270               ;END ANGLE C AXIS ~
    Q421=+0                 ;INCID. ANGLE C AXIS ~
    Q422=+0                 ;MEAS. POINTS C AXIS ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q432=+0                 ;BACKLASH, ANG. RANGE
```

The goal of this procedure is that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, the adjustment of a fork head with A and C axes is described. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- Insert the interchangeable head that will be used as a reference head.
- Clamp the calibration sphere
- Insert the touch probe
- Use Cycle **451** to measure the complete kinematics, including the reference head
- Define the preset (using **Q431** = 2 or 3 in Cycle **451**) after measuring the reference head

Measuring a reference head

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 451 MEASURE KINEMATICS ~
    Q406=+1                 ;MODE ~
    Q407=+12.5              ;SPHERE RADIUS ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q408=+0                 ;RETR. HEIGHT ~
    Q253=+2000              ;F PRE-POSITIONING ~
    Q380=+45                ;REFERENCE ANGLE ~
    Q411=-90                ;START ANGLE A AXIS ~
    Q412=+90                ;END ANGLE A AXIS ~
    Q413=+45                ;INCID. ANGLE A AXIS ~
    Q414=+4                 ;MEAS. POINTS A AXIS ~
    Q415=-90                ;START ANGLE B AXIS ~
    Q416=+90                ;END ANGLE B AXIS ~
    Q417=+0                 ;INCID. ANGLE B AXIS ~
    Q418=+2                 ;MEAS. POINTS B AXIS ~
    Q419=+90                ;START ANGLE C AXIS ~
    Q420=+270               ;END ANGLE C AXIS ~
    Q421=+0                 ;INCID. ANGLE C AXIS ~
    Q422=+3                 ;MEAS. POINTS C AXIS ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q431=+3                 ;PRESET ~
    Q432=+0                 ;BACKLASH, ANG. RANGE
```

Drift compensation

This procedure can also be performed on machines without rotary axes.

During machining, various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle **452**.

- Clamp the calibration sphere
- Insert the touch probe
- Measure the complete kinematics with Cycle **451** before starting the machining process
- Define the preset (using **Q432** = 2 or 3 in Cycle **451**) after measuring the kinematics
- Then set the presets on your workpiece and start the machining process.

Reference measurement for drift compensation

```
11 TOOL CALL "TOUCH_PROBE" Z
12 CYCL DEF 247 PRESETTING ~
   Q339=+1              ;PRESET NUMBER
13 TCH PROBE 451 MEASURE KINEMATICS ~
   Q406=+1              ;MODE ~
   Q407=+12.5           ;SPHERE RADIUS ~
   Q320=+0              ;SET-UP CLEARANCE ~
   Q408=+0              ;RETR. HEIGHT ~
   Q253=+750            ;F PRE-POSITIONING ~
   Q380=+45             ;REFERENCE ANGLE ~
   Q411=+90             ;START ANGLE A AXIS ~
   Q412=+270            ;END ANGLE A AXIS ~
   Q413=+45             ;INCID. ANGLE A AXIS ~
   Q414=+4              ;MEAS. POINTS A AXIS ~
   Q415=-90             ;START ANGLE B AXIS ~
   Q416=+90             ;END ANGLE B AXIS ~
   Q417=+0              ;INCID. ANGLE B AXIS ~
   Q418=+2              ;MEAS. POINTS B AXIS ~
   Q419=+90             ;START ANGLE C AXIS ~
   Q420=+270            ;END ANGLE C AXIS ~
   Q421=+0              ;INCID. ANGLE C AXIS ~
   Q422=+3              ;MEAS. POINTS C AXIS ~
   Q423=+4              ;NO. OF PROBE POINTS ~
   Q431=+3              ;PRESET ~
   Q432=+0              ;BACKLASH, ANG. RANGE
```

- Measure the drift of the axes at regular intervals.
- Insert the touch probe
- Activate the preset in the calibration sphere.
- Use Cycle **452** to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the entire process.

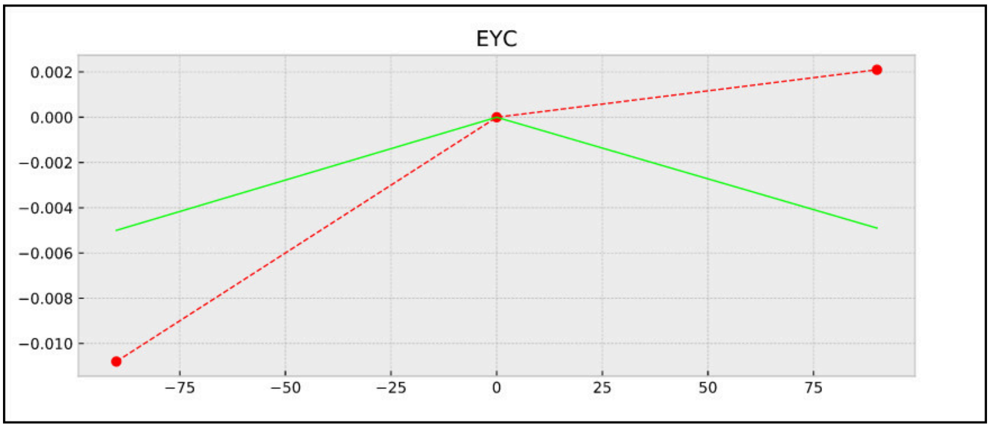
Drift compensation

```
11 TOOL CALL "TOUCH_PROBE" Z
13 TCH PROBE 452 PRESET COMPENSATION ~
    Q407=+12.5          ;SPHERE RADIUS ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q408=+0             ;RETR. HEIGHT ~
    Q253=+9999          ;F PRE-POSITIONING ~
    Q380=+45            ;REFERENCE ANGLE ~
    Q411=-90            ;START ANGLE A AXIS ~
    Q412=+90            ;END ANGLE A AXIS ~
    Q413=+45            ;INCID. ANGLE A AXIS ~
    Q414=+4             ;MEAS. POINTS A AXIS ~
    Q415=-90            ;START ANGLE B AXIS ~
    Q416=+90            ;END ANGLE B AXIS ~
    Q417=+0             ;INCID. ANGLE B AXIS ~
    Q418=+2             ;MEAS. POINTS B AXIS ~
    Q419=+90            ;START ANGLE C AXIS ~
    Q420=+270           ;END ANGLE C AXIS ~
    Q421=+0             ;INCID. ANGLE C AXIS ~
    Q422=+3             ;MEAS. POINTS C AXIS ~
    Q423=+3             ;NO. OF PROBE POINTS ~
    Q432=+0             ;BACKLASH, ANG. RANGE
```

Log function

After running Cycle **452**, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Tool name
- Active kinematics
- Mode used
- Inclination angles
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Number of measuring points
  - Measuring circle radius
  - Averaged backlash, if **Q423>0**
  - Positions of the axes
  - Standard deviation (scatter)
  - Maximum deviation
  - Angular error
  - Compensation values in all axes (preset shift)
  - Position before preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Averaged positioning error
  - SVG files with graphs: measured and optimized errors of individual measurement positions.
    - Red curve: measured positions
    - Green curve: optimized values
    - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = deviations) of the Y axis in dependency of the C axis.
    - X axis of the graph: rotary axis position in degrees
    - Y axis of the graph: position deviations in mm



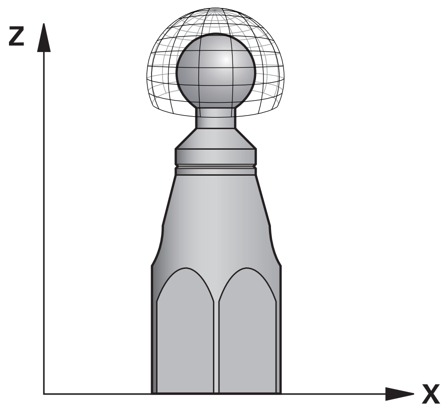
Sample measurement: EYC deviations of the Y axis in dependency of the C axis

31.7.5 Cycle 453 KINEMATICS GRID

ISO programming
G453

Application

	Refer to your machine manual. KinematicsOpt (software option 48) is required. This function must be enabled and adapted by the machine manufacturer. To use this cycle, your machine manufacturer needs to create and configure a compensation table (*.kco) first and enter some more settings.



Even if your machine was already optimized regarding positioning errors (e.g., via Cycle **451**), residual errors at the Tool Center Point (**TCP**) during tilting of the rotary axes may remain. These can result, for example, from component errors (e.g., a bearing error) with head rotation axes.

Cycle **453 KINEMATICS GRID** enables errors in swivel heads to be detected and compensated for in accordance with the rotary axis positions. If you want to write compensation values with this cycle, then you need **KinematicsComp** (option 52). With this cycle and using a 3D TS touch probe, you measure a HEIDENHAIN calibration sphere that you have attached to the machine table. The cycle then moves the touch probe automatically to positions in a grid-line arrangement around the calibration sphere. The machine manufacturer defines these swivel axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing operation on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine manufacturer defines this compensation table (*.kco) and specifies its storage location.

When using Cycle **453**, run it at different positions in the workspace. This allows you to check immediately if compensation with Cycle **453** has the desired positive effect on the machine's accuracy. Only when the desired improvements are achieved with the same compensation values at several positions is such a type of compensation suitable for the respective machine. If this is not the case, then the errors are to be sought outside the rotary axes.

Perform the measurement with Cycle **453** in an optimized condition regarding the rotary axis positioning errors. For this purpose, use e.g. Cycle **451** beforehand.

HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The control then optimizes the accuracy of your machine. For this purpose, it automatically saves the compensation values resulting from a measurement in a compensation table (*.kco). (This applies to mode **Q406**=1.)

Cycle sequence

1 Clamp the calibration sphere and check for potential collisions.
2 In Manual mode of operation, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select one of the Program Run operating modes and start the NC program
4 The cycle is executed in accordance with the setting in **Q406** (–1=Delete mode / 0=Test mode / 1=Compensate mode)

> During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

Various modes (Q406)

Deletion mode Q406 = -1 (KinematicsComp, option 52)

- The axes are not moved
- The control writes all values to the compensation table (*.kco), setting them to "0". The result is that no further compensations will be effective for the currently selected kinematics.

Test mode Q406 = 0

- The control probes the calibration sphere.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Compensation mode Q406 = 1 (KinematicsComp, option 52)

- The control probes the calibration sphere.
- The control writes the deviations to the compensation table (*.kco). The table is updated and the compensation settings are immediately effective.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. It is recommended to clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Notes

> KinematicsOpt (software option 48) is required. KinematicsComp (software option 52) is required.
>
> This function must be enabled and adapted by the machine manufacturer.
>
> Your machine manufacturer defines the storage location of the compensation table (*.kco).

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- Deactivate the basic rotation before running the cycle.
- Set the preset and the basic rotation again after optimization.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or you set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.
- If you have activated preset setting before the calibration (**Q431** = 1/3), then move the touch probe by the amount of the set-up clearance (**Q320** + **SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.

- If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.

Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the maximum permissible modification of a transformation. If the value is not equal to –1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	Q406 Mode (-1/0/+1) Define whether the control will write a value of 0 to the values of the compensation table (*.kco), will check the currently existing deviations, or will perform a compensation. A log file (*.html) is created. **–1**: Delete values in the compensation table (*.kco). The compensation values for TCP positioning errors are set to 0 in the compensation table (*.kco). The control will not perform any probing. No results will be output to the log (*.html). (requires **KinematicsComp**, option 52) **0**: Check TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions but does not write values to the compensation table (*.kco). The control displays the standard and maximum deviation in a log (*.html). **1**: Compensate for TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions and writes the deviations to the compensation table (*.kco). The compensations are then immediately effective. The control displays the standard and maximum deviation in a log (*.html). (requires **KinematicsComp**, option 52) Input: **-1**, **0**, **+1**
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	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** or **PREDEF**
	Q408 Retraction height? **0**: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. **> 0**: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**. This value has an absolute effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: **0...360**
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision. Input: **3...8**
	Q431 Preset (0/1/2/3)? Define whether the control will automatically set the active preset at the center of the sphere: **0**: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle **1**: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle **2**: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle **3**: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle Input: **0**, **1**, **2**, **3**

Probing with Cycle 453

```
11 TCH PROBE 453 KINEMATICS GRID ~
    Q406=+0          ;MODE ~
    Q407=+12.5       ;SPHERE RADIUS ~
    Q320=+0          ;SET-UP CLEARANCE ~
    Q408=+0          ;RETR. HEIGHT ~
    Q253=+750        ;F PRE-POSITIONING ~
    Q380=+0          ;REFERENCE ANGLE ~
    Q423=+4          ;NO. OF PROBE POINTS ~
    Q431=+0          ;PRESET
```

Log function

After running Cycle **453**, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder where the current NC program resides. It contains the following data:

- Date and time of protocol creation
- Path of the NC program from which the cycle was run
- Number and name of the currently active tool
- Mode
- Measured data: Standard deviation and maximum deviation
- Information at which position in degrees (°) the maximum deviation occurred
- Number of measuring positions

31.8 Touch Probe Cycles: Automatic Tool Measurement

31.8.1 Fundamentals

Overview

Refer to your machine manual.

Some cycles and functions may not be provided on your machine.

Option 17 is required.

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.

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 **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

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

Cycle		Call	Further information
480 **30**	**CALIBRATE TT** ■ Calibrating the tool touch probe	**DEF**-active	Page 1910
481 **31**	**CAL. TOOL LENGTH** ■ Measuring the tool length	**DEF**-active	Page 1913
482 **32**	**CAL. TOOL RADIUS** ■ Measuring the tool radius	**DEF**-active	Page 1917
483 **33**	**MEASURE TOOL** ■ Measuring the tool length and radius	**DEF**-active	Page 1920
484	**CALIBRATE IR TT** ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	**DEF**-active	Page 1924
485	**MEASURE LATHE TOOL** (option 50) ■ Measurement of turning tools	**DEF**-active	Page 1928

Differences between Cycles 30 to 33 and Cycles 480 to 483

The features and the operating sequences are absolutely identical. There are only the following differences between Cycles **30** to **33** and Cycles **480** to **483**:

- Cycles **480** to **483** are also available as **G480** to **G483** for ISO programming
- Instead of a selectable parameter for the status of the measurement, Cycles **481** to **483** use the fixed parameter **Q199**.

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).

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius [mm]

The probing feed rate is calculated as follows:
v = measuring tolerance • n with

v:	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate:

probingFeedCalc (no. 122710) = **ConstantTolerance**:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed **maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance **measureTolerance1** (no. 122715), the sooner you will encounter this effect.

probingFeedCalc (no. 122710) = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc (No. 122710) = **ConstantFeed**:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance = (r • **measureTolerance1**)/ 5 mm) where

r:	Active tool radius [mm]
measureTolerance1:	Maximum permissible error of measurement

Entries in the tool table for milling and turning tools

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0.0000 to 5.0000 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0.0000 to 5.0000 mm	**Wear tolerance: radius?**
DIRECT.	Cutting direction of the tool for measuring a rotating tool	**Cutting direction (M3 = -)?**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to **offsetToolAxis**. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0.0000 to 9.0000 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0.0000 to 9.0000 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	No function	0: No offset required because tool tip is to be measured	
End mill	4: 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.

31.8.2 Cycle 30 or 480 CALIBRATE TT

ISO programming
G480

Application

	Refer to your machine manual!

You calibrate the TT with touch probe cycle **30** or **480** (Page 1907). The calibration process runs automatically. The control also measures the center misalignment 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 **30** or **480** .

Touch probe

For the touch probe you use a spherical or cuboid probe contact

Cuboid probe contact

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined. Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

Further information: "TT tab", Page 183

	When clamping the tool touch probe, make sure that the edges of the cuboid probe contact are aligned as parallel to the machine axes as possible. The angle of misalignment should be less than 1° and the tilt angle should be less than 0.3°.

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 sequence

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**

Example of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 480 CALIBRATE TT ~
    Q260=+100                    ;CLEARANCE HEIGHT
```

Example of old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 30.0 CALIBRATE TT
13 TCH PROBE 30.1 HEIGHT:+90
```

31.8.3 Cycle 31 or 481 CAL. TOOL LENGTH

ISO programming
G481

Application

⚙	Refer to your machine manual!

If you want to measure the tool length, program the touch probe cycle **31** or **482** (Page 1907). Input parameters allow you to select which of the three following methods will be used to measure the tool length:

- 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. To activate this function, set the parameter **PROBING THE TEETH** = 1 in Cycle **31**.

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.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**LBREAK** and **LTOL**) from the **TOOL.T** table.

Q340: **0** and **1**

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools, see "Tool data", Page 275.

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** Note the behavior with grinding tools, **Further information:** "Measuring grinding tools", Page 1914
	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 of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 481 CAL. TOOL LENGTH ~
    Q340=+1              ;CHECK ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q341=+1              ;PROBING THE TEETH
```

Cycle **31** includes an additional parameter:

Help graphic	Parameter
	Parameter number for result? Parameter number in which the control stores the status of the measurement: **0.0**: Tool is within the tolerance **1.0**: Tool is worn (**LTOL** exceeded) **2.0**: Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with **NO ENT** Input: **0...1999**

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:0
14 TCH PROBE 31.2 HEIGHT::+120
15 TCH PROBE 31.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:1 Q5
14 TCH PROBE 31.2 HEIGHT:+120
15 TCH PROBE 31.3 PROBING THE TEETH:1
```

31.8.4 Cycle 32 or 482 CAL. TOOL RADIUS

ISO programming
G482

Application

⚙	Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **32** or **482** (Page 1907). Input parameters allow you to select which of the two following methods will be used to measure the tool radius:

- 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.

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.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**RBREAK** and **RTOL**) from the **TOOL.T** table.

Q340: **0** and **1**

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools

Further information: "Tool data for the tool types", Page 284

Notes about 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.

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 of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 482 CAL. TOOL RADIUS ~
    Q340=+1                 ;CHECK ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q341=+1                 ;PROBING THE TEETH
```

Cycle **32** includes an additional parameter:

Help graphic	Parameter
	Parameter number for result? Parameter number in which the control stores the status of the measurement: **0.0**: Tool is within the tolerance **1.0**: Tool is worn (**RTOL** exceeded) **2.0**: Tool is broken (**RBREAK** exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with **NO ENT** Input: **0...1999**

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:0
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:1 Q5
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:1
```

31.8.5 Cycle 33 or 483 MEASURE TOOL

ISO programming

G483

Application

Refer to your machine manual!

To measure both the length and radius of a tool, program the touch probe cycle **33** or **483** (Page 1907). 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 **31** and **32** as well as **481** and **482**.

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.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**LBREAK** , **RBREAK**, **LTOL**, and **RTOL**) from the **TOOL.T** table.

Q340: **0** and **1**

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools

Further information: "Tool data for the tool types", Page 284

Notes about 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.

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 of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 483 MEASURE TOOL ~
    Q340=+1                 ;CHECK ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q341=+1                 ;PROBING THE TEETH
```

Cycle **33** includes an additional parameter:

Help graphic	Parameter
	Parameter number for result? Parameter number in which the control stores the status of the measurement: **0.0**: Tool is within the tolerance **1.0**: Tool is worn (**LTOL** or/and **RTOL** exceeded) **2.0**: Tool is broken (**LBREAK** or/and **RBREAK** exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with **NO ENT**. Input: **0...1999**

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:0
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:1 Q5
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:1
```

31.8.6 Cycle 484 CALIBRATE IR TT

ISO programming
G484

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 sequence

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 or cuboid probe contact

Cuboid probe contact:

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined. Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

Further information: "TT tab", Page 183

When clamping the tool touch probe, make sure that the edges of the cuboid probe contact are aligned as parallel to the machine axes as possible. The angle of misalignment should be less than 1° and the tilt angle should be less than 0.3°.

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.
- ▶ 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 sequence
- > 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 the **CANCEL** button 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**

Example

```
11 TOOL CALL 12 Z
12 TCH PROBE 484 CALIBRATE IR TT ~
    Q536=+0                 ;STOP BEFORE RUNNING ~
    Q523=+0                 ;TT POSITION
```

31.8.7 Cycle 485 MEASURE LATHE TOOL (option 50)

ISO programming
G485

Application

> Refer to your machine manual!
> Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

Cycle **485 MEASURE LATHE TOOL** is available for the measurement of lathe tools using the tool touch probe from HEIDENHAIN. The control measures the tool in a fixed programmed sequence.

Cycle sequence

1. The control positions the lathe tool to the clearance height
2. The lathe tool is oriented based on the entries in **TO** and **ORI**
3. The control moves the tool to the measuring position in the main axis; traverse movement is interpolated in the main and secondary axes
4. Then the lathe tool moves to the measuring position in the tool axis
5. The tool is measured. Depending on the definition of **Q340**, either tool dimensions are changed or the tool is locked
6. The measuring result is transferred to the result parameter **Q199**
7. After the measurement has been performed, the control positions the tool in the tool axis to the clearance height

Result parameter Q199:

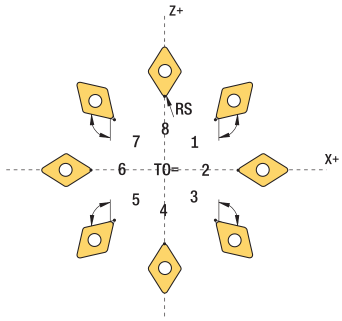
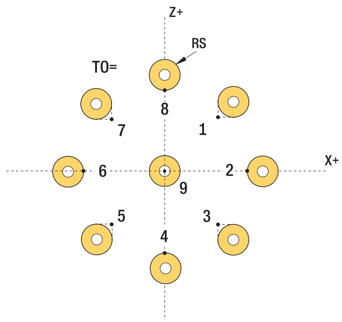
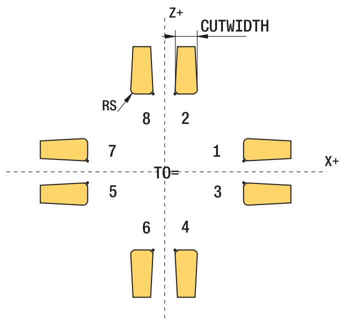
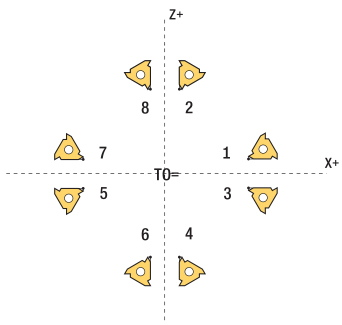
Result	Meaning
0	Tool dimensions within the tolerance **LTOL** / **RTOL** Tool is not locked
1	Tool dimensions outside the tolerance **LTOL** / **RTOL** Tool is locked
2	Tool dimensions outside the tolerance **LBREAK** / **RBREAK** Tool is locked

The cycle uses the following entries from toolturn.trn:

Abbr.	Entries	Dialog
ZL	Tool length 1 (**Z** direction)	**Tool length 1?**
XL	Tool length 2 (**X** direction)	**Tool length 2?**
DZL	Delta value of tool length 1 (**Z** direction), is added to **ZL**	**Oversize in tool length 1?**
DXL	Delta value of tool length 2 (**X** direction), is added to **XL**	**Oversize in tool length 2?**
RS	Cutting edge radius: if contours were programmed with radius compensation **RL** or **RR**, the control takes the cutting edge radius into account in turning cycles, and performs cutting radius compensation	**Cutting edge radius?**
TO	Tool orientation: from the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction, position of the tool reference point, etc. This information is necessary, for example, for calculating the cutting radius compensation, milling cutter radius compensation, plunge angle, etc.	**Tool orientation?**
ORI	Spindle orientation angle: angle of the indexable insert to the main axis	**Angle of spindle orientation?**
TYPE	Type of turning tool: Roughing tool **ROUGH**, finishing tool **FINISH**, thread tool **THREAD**, recessing tool **RECESS**, button tool **BUTTON**, groove turning tool **RECTURN**	**Type of turning tool**

Further information: "Tool orientation (TO) that is supported for the following types of turning tools (TYPE)", Page 1930

Tool orientation (TO) that is supported for the following types of turning tools (TYPE)

TYPE	Supported TO with possible limitations	Non-supported TO	
ROUGH, FINISH	■ 1 ■ 7 ■ 2, only **XL** ■ 3, only **XL** ■ 5, only **XL** ■ 6, only **XL** ■ 8, only **ZL** ■ 18	■ 4 ■ 9	
BUTTON	■ 1 ■ 7 ■ 2, only **XL** ■ 3, only **XL** ■ 5, only **XL** ■ 6, only **XL** ■ 8, only **ZL**	■ 4 ■ 9	
RECESS, RECTURN	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only **XL** ■ 5, only **XL**	■ 4 ■ 6 ■ 9	
THREAD	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only **XL** ■ 5, only **XL**	■ 4 ■ 6 ■ 9	

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

NOTICE

Danger of collision!

If the tool data **ZL** / **DZL** and **XL** / **DXL** deviate by more than ±2 mm from the real tool data, then there is a danger of collision.

- Enter the approximate tool data closer than ±2 mm
- Run the cycle carefully

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before you begin the cycle, you must run a **TOOL CALL** with the tool axis **Z**.
- If you define **YL** and **DYL** with a value outside of ±5 mm, the tool won't reach tool touch probe.
- The cycle does not support **SPB-INSERT** (angular offset). You must enter the value 0 in **SPB-INSERT**, otherwise the control will generate an error message.

Note regarding machine parameters

- The cycle depends on the optional machine parameter **CfgTTRectStylus** (no. 114300). Refer to your machine manual.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Use of the measured values: **0**: The measured values are entered in **ZL** and **XL**. If values are already entered in the tool table, they will be overwritten. **DZL** and **DXL** will be reset to **0**. TL will not be changed **1**: The measured values **ZL** and **XL** are compared with the values from the tool table. These values will not be changed. The control then calculates the deviations of **ZL** and **XL**, and enters these in **DZL** and **DXL**. If the delta values are larger than the permissible wear or breakage tolerance, the control locks the tool (**TL** = Tool Locked). In addition, the deviation is also entered in the Q parameters **Q115** and **Q116** **2**: The measured values **ZL** and **XL** as well as **DZL** and **DXL** are compared with the values from the tool table, but are not changed. If the values are larger than the permissible wear or breakage tolerance, the control locks the tool (**TL** = Tool Locked). 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**

Example

```
11 TOOL CALL 12 Z
12 TCH PROBE 485 MEASURE LATHE TOOL ~
    Q340=+1                   ;CHECK ~
    Q260=+100                 ;CLEARANCE HEIGHT
```

32

The MDI Application

Application

The **MDI** application allows you to execute individual NC blocks outside of the context of an NC program (e.g., **PLANE RESET**). When you press the **NC Start** key, the control will run the NC blocks separately.

You can also create an NC program step by step. The control remembers modally effective program information.

Related topics

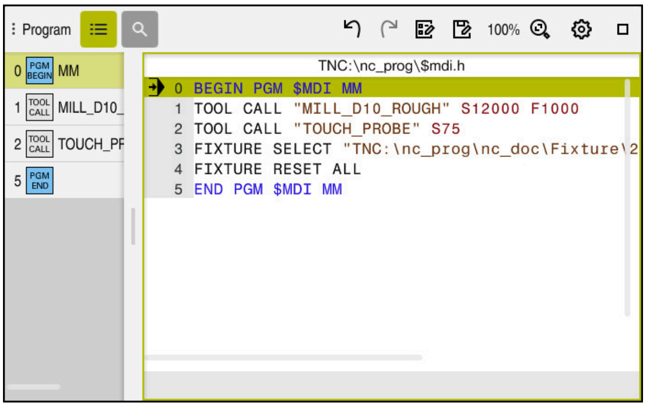
- Creating NC programs

  Further information: "Programming fundamentals", Page 212
- Running NC programs

  Further information: "Program Run", Page 1953

Description of function

If you program using the millimeter unit of measurement, the control will use the NC program **$mdi.h** by default. If you program using the inch unit of measurement, the control will use the NC program **$mdi_inch.h**.



Program workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **GS** (option 44)

  Further information: "Global Program Settings (GPS, option 44)", Page 1217
- **Help**
- **Positions**

  Further information: "Positions workspace", Page 163
- **Program**

  Further information: "Program workspace", Page 217
- **Simulation**

  Further information: "Simulation Workspace", Page 1535
- **Status**

  Further information: "Status workspace", Page 171
- **Keyboard**

  Further information: "Virtual keyboard of the control bar", Page 1508

Buttons

In the **MDI** application, the function bar provides the following buttons:

Button	Meaning
Klartext programming	If this toggle switch is active, then you are using dialog-guided programming. If this toggle switch is not active, then you are programming in the text editor. **Further information:** "Editing NC programs", Page 228
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Inserting NC functions", Page 228
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1366
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** "GOTO function", Page 1511
/ Skip block off/on	Hide NC blocks with a **/** character. NC blocks hidden with a **/** character will be ignored during program run as soon as the **Skip block** toggle switch is active. **Further information:** "Hiding NC blocks", Page 1513
Skip block	If this toggle switch is active, the control ignores NC blocks hidden with a **/** character. **Further information:** "Hiding NC blocks", Page 1513 If the toggle switch is active, the control grays out the NC blocks to be skipped. **Further information:** "Appearance of the NC program", Page 219
; Comment Off/On	Insert or remove a **;** character in front of an NC block. If an NC block begins with a **;** character, then the block is a comment. **Further information:** "Adding comments", Page 1512
FMAX	You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit FMAX", Page 1958
F limited	You activate or deactivate the feed-rate limitation for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 2096
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC, option 145). **Further information:** "Active Chatter Control (ACC, option 145)", Page 1204
Editing	The control opens the context menu. **Further information:** "Context menu", Page 1522
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Interrupting, stopping or canceling program run", Page 1959
Reset program	If you select **Internal stop**, the control activates this button. The control places the cursor back to the beginning of the program and resets any modally effective program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always run the NC blocks in **Single Block** mode. After the control has run an NC block, the program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 1959

The block numbers of all NC blocks that you have successively run are shown in green.

The control saves the following data in this state:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

Notes

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the window **Q parameter list**
  - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

- In the **MDI** application, you can create and execute NC programs step by step. Then you can use **Save as** to save the current contents with a different file name.
- The following functions are not available in the **MDI** application:
  - Calling an NC program with **PGM CALL**, **SEL PGM**, or **CALL SELECTED PGM**
  - Test Run in the **Simulation** workspace
  - **Manual traverse** and **Approach position** while program run is interrupted
  - **Block scan** function

33

Pallet Machining and Job Lists

33.1 Fundamentals

⚙	Refer to your machine manual. Pallet table management is a machine-dependent function. The standard functional range is described below.

Pallet tables (**.p**) are mainly used in machining centers with pallet changers. The pallet tables call the different pallets (PAL), fixtures (FIX) optionally, and the associated NC programs (PGM). The pallet tables activate all defined presets and datum tables.

Without a pallet changer, you can use pallet tables to successively run NC programs with different presets with just one press of **NC Start**. This type of usage is also called job list.

Tool-oriented machining is possible with pallet tables and with job lists. The control will reduce the number of tool changes, thereby reducing the machining time.

Further information: "Tool-oriented machining", Page 1948

33.1.1 Pallet counter

You can define a pallet counter on the control. This allows you to define a variable number of parts produced, for example during pallet machining with automatic workpiece change.

For this purpose, you define a value in the **TARGET** column of the pallet table. The control repeats the NC programs of this pallet until the nominal value has been reached.

By default, every NC program that has been executed increases the actual value by 1. If, for example, an NC program produces more than one workpiece, you define the value in the **COUNT** column of the pallet table.

Further information: "Pallet table", Page 2051

The control displays the defined nominal value and the current actual value in the **Job list** workspace.

Further information: "Information about the pallet table", Page 1939

33.2 Job list workspace

33.2.1 Fundamentals

Application

In the **Job list** workspace, you edit and execute pallet tables.

Related topics

- Contents of a pallet table

  Further information: "Pallet table", Page 2051
- **Form** workspace for pallets

  Further information: "Form workspace for pallets", Page 1946
- Tool-oriented machining

  Further information: "Tool-oriented machining", Page 1948

Description of function

In the **Job list** workspace, the control displays the individual rows of the pallet table and the status.

Further information: "Information about the pallet table", Page 1939

If you activate the **Edit** toggle switch, the **Insert row** button will be displayed in the action bar and allows you to insert a new table row.

Further information: "Insert row window", Page 1941

When you open a pallet table in **Editor** or **Program Run** operating mode, the control will automatically display the **Job list** workspace. You cannot close this workspace.

Information about the pallet table

When you open a pallet table, the following information will be displayed in the **Job list** workspace:

Column	Meaning
No column name	Status of the pallet, fixture, or NC program In the **Program Run** operating mode: execution cursor **Further information:** "Status of the pallet, fixture, or NC program", Page 1939
Program	Information about the pallet counter: ■ For rows of the **PAL** type: Current actual value (**COUNT**) and defined nominal value (**TARGET**) of the pallet counter. ■ For rows of the **PGM** type: Value indicating by how much the actual value will be incremented after the execution of the NC program. **Further information:** "Pallet counter", Page 1938 Machining method: ■ Workpiece-oriented machining ■ Tool-oriented machining **Further information:** "Machining method", Page 1940
Sts	Machining status **Further information:** "Machining status", Page 1940

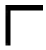
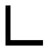
Status of the pallet, fixture, or NC program

The control uses the following icons to display the status:

Icon	Meaning
	The **Pallet**, **Clamping**, or **Program** is locked
	The **Pallet** or **Clamping** is not enabled for machining
	This line is currently being executed in **Program run, single block** or **Program run, full sequence** mode and cannot be edited
	In this line, the program was interrupted manually

Machining method

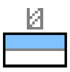
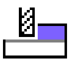
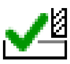
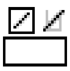
The control uses the following icons to display the machining method:

Icon	Meaning
No icon	Workpiece-oriented machining
 	Tool-oriented machining ■ Start ■ End

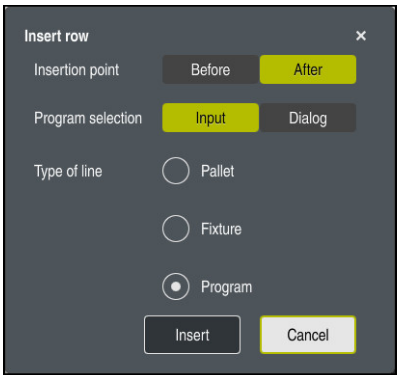
Machining status

The control updates the machining status during program run.

The control uses the following icons to display the machining status:

Icon	Meaning
	Workpiece blank, machining required
	Partially machined, requires further machining
	Completely machined, no further machining required
	Skip machining

Insert row window



Insert row window with **Program** selected

The **Insert row** window provides the following settings:

Setting	Meaning
Insertion point	■ **Before**: Insert a new row before the current cursor position ■ **After**: Insert a new row after the current cursor position
Program selection	■ **Input**: Enter the path of the NC program ■ **Dialog**: Select the NC program via a selection window
Type of line	Corresponds to the **TYPE** column of the pallet table Insert a **Pallet**, **Clamping** or **Program**

You can edit the contents and settings of a row in the **Form** workspace.

Further information: "Form workspace for pallets", Page 1946

Program Run operating mode

You can open the **Program** workspace in addition to the **Job list** workspace. After you have selected a table row with an NC program, the control displays the program contents in the **Program** workspace.

The control uses the execution cursor to indicate which table row is marked for running or is currently being run.

Use the **GOTO Cursor** button to move the execution cursor to the currently selected row of the pallet table.

Further information: "Mid-program startup at any NC block", Page 1942

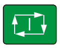
Mid-program startup at any NC block

To perform a block scan for mid-program startup at an NC block:

- Open the pallet table in **Program Run** operating mode
- Open the **Program** workspace
- Select the table row with the desired NC program

- Select **GOTO Cursor**
- > The control marks the table row with the execution cursor.
- > The control displays the contents of the NC program in the **Program** workspace.
- Select the desired NC block

- Select **Block scan**
- > The control opens the **Block scan** window displaying the values of the NC block.



- Press the **NC Start** key
- > The control starts the block scan.

Notes

- After you have opened a pallet table in **Program Run** operating mode, you can no longer edit this pallet table in **Editor** operating mode.
- In the machine parameter **editTableWhileRun** (no. 202102), the machine manufacturer defines whether you will be allowed to edit the pallet table during program run.
- In the machine parameter **stopAt** (no. 202101), the machine manufacturer defines when the control will stop program run during the execution of a pallet table.
- In the optional machine parameter **resumePallet** (no. 200603), the machine manufacturer defines whether the control will continue program execution after an error message.
- The optional machine parameter **failedCheckReact** (no. 202106) allows you to define whether the control checks incorrect tool or program calls.
- The optional machine parameter **failedCheckImpact** (no. 202107) allows you to define whether the control skips the NC program, the fixture or the pallet after an incorrect tool or program call.

33.2.2 Batch Process Manager (option 154)

Application

Batch Process Manager enables you to plan production orders on a machine tool.

The Batch Process Manager software option allows the control to display the following additional information in the **Job list** workspace:

- Times at which manual interventions at the machine are necessary
- Run time of the NC programs
- Availability of the tools
- Whether the NC program is free of errors

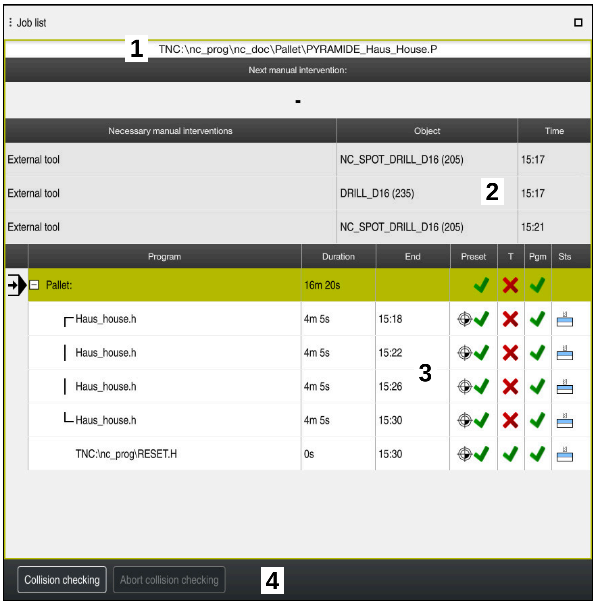
Related topics

- **Job list** workspace
  Further information: "Job list workspace", Page 1938
- Editing a pallet table in the **Form** workspace
  Further information: "Form workspace for pallets", Page 1946
- Contents of the pallet table
  Further information: "Pallet table", Page 2051

Requirements

- Pallet Management (software option 22)
- Batch Process Manager (software option 154)
  Batch Process Manager is an expansion to the pallet management feature. Batch Process Manager provides you with all functions available in the **Job list** workspace.
- Tool usage test is active
  The tool usage test function has to be enabled and switched on to ensure you get all information!
  Further information: "Channel settings", Page 2104

Description of function



Job list workspace with **Batch Process Manager** (option 154)

When Batch Process Manager is enabled, the **Job list** workspace provides the following areas:

1 File information bar
   In the file information bar, the control shows the path of the pallet table.
2 Information about necessary manual interventions
   - Time until the next manual intervention
   - Type of intervention
   - Affected object
   - Time of manual intervention
3 Information about and status of the pallet table
   Further information: "Information about the pallet table", Page 1945
4 Action bar
   If the **Edit** toggle switch is active, you can add a new row.
   If the **Edit** toggle switch is inactive, you can use the Dynamic Collision Monitoring (DCM) feature (option 40) to check all NC programs of the pallet table in **Program Run** operating mode.

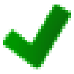
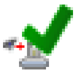
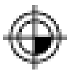
Information about the pallet table

When you open a pallet table, the following information is displayed in the **Job list** workspace:

Column	Meaning
No column name	Status of the pallet, fixture, or NC program In the **Program Run** operating mode: execution cursor **Further information:** "Status of the pallet, fixture, or NC program", Page 1939
Program	Name of the pallet, fixture, or NC program Information about the pallet counter: ■ For rows of the **PAL** type: Current actual value (**COUNT**) and defined nominal value (**TARGET**) of the pallet counter. ■ For rows of the **PGM** type: Value indicating by how much the actual value will be incremented after the execution of the NC program. **Further information:** "Pallet counter", Page 1938 Machining method: ■ Workpiece-oriented machining ■ Tool-oriented machining **Further information:** "Machining method", Page 1940
Duration	Duration of executing the pallet, fixture, or NC program
End	Expected point in time after execution of the NC program In the **Editor** operating mode the **End** column does not show a point in time but the duration.
Preset	Status of the workpiece preset: ■ Workpiece preset is defined ■ Check input **Further information:** "Status of the workpiece preset, the tools, and the NC program", Page 1946
T	Status of the tools used: ■ Test completed ■ Test not yet completed ■ Test failed The column only shows the status in the **Program Run** operating mode. **Further information:** "Status of the workpiece preset, the tools, and the NC program", Page 1946
Pgm	Status of the NC program: ■ Test completed ■ Test not yet completed ■ Test failed **Further information:** "Status of the workpiece preset, the tools, and the NC program", Page 1946
Sts	Machining status **Further information:** "Machining status", Page 1940

Status of the workpiece preset, the tools, and the NC program

The control displays the status using the following icons:

Icon	Meaning
	Test completed
	Test completed Program simulation with active **Dynamic Collision Monitoring (DCM)** (option 40)
	Test failed (e.g., because of expired tool life, danger of collision)
	Test not yet completed
	Incorrect program structure (e.g., pallet does not contain any subprograms)
	Workpiece preset is defined
	Check input You can assign a workpiece preset either to the pallet or to all NC subprograms.

Note

If you edit the job list, the Collision checking completed  status is reset to Check completed .

33.3 Form workspace for pallets

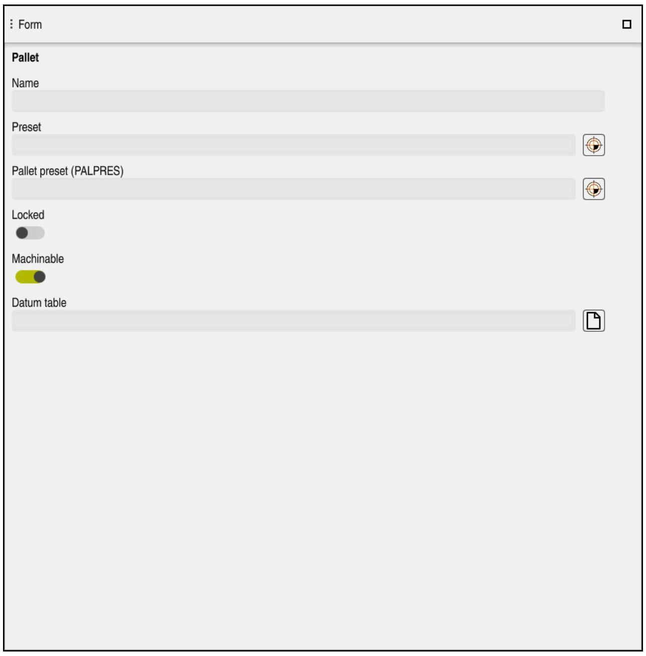
Application

In the **Form** workspace, the control shows the contents of the pallet table for the selected row.

Related topics

- **Job list** workspace
  Further information: "Job list workspace", Page 1938
- Contents of the pallet table
  Further information: "Pallet table", Page 2051
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 1948

Description of function



Form workspace with the contents of a pallet table

A pallet table can have the following types of rows:

- **Pallet**
- **Clamping**
- **Program**

In the **Form** workspace, the control shows the contents of the pallet table. The control shows the contents relevant to the respective type of the selected row.

You can edit the settings in the **Form** workspace or in **Tables** operating mode. The control synchronizes the contents.

By default, the names of the table columns are used to designate the settings options in the form.

The toggle switches provided in the form correspond to the following table columns:

- The **Locked** toggle switch corresponds to the column **LOCK**
- The **Machinable** toggle switch corresponds to the column **LOCATION**

If the control displays an icon next to the input field, a selection window for selecting the contents is available

The **Form** workspace can be selected for pallet tables in **Editor** or **Program Run** operating mode.

33.4 Tool-oriented machining

Application

Tool-oriented machining allows you to machine several workpieces together even on a machine without pallet changer, which reduces tool-change times. You can thus use the pallet management feature even on machines without a pallet changer.

Related topics

- Contents of the pallet table

  Further information: "Pallet table", Page 2051
- Block scan for mid-program startup in a pallet table

  Further information: "Block scan in pallet tables", Page 1971

Requirements

- Pallet Management (software option 22)
- Tool-change macro for tool-oriented machining
- **METHOD** column with the values **TO** or **TCO**
- NC programs with identical tools

  The tools being used must, at least in part, be the same tools.
- **W-STATUS** column with the values **BLANK** or **INCOMPLETE**
- NC programs must not contain the following functions:
  - **FUNCTION TCPM** or **M128** (option 9)

    Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
  - **M144** (option 9)

    Further information: "Factoring the tool offset into the calculations with M144 (option 9)", Page 1350
  - **M101**

    Further information: "Automatically inserting a replacement tool with M101", Page 1355
  - **M118**

    Further information: "Activating handwheel superimpositioning with M118", Page 1334
  - Changing the pallet preset

    Further information: "Pallet preset table", Page 1951

Description of function

The following columns of the pallet table apply to tool-oriented machining:

- **W-STATUS**
- **METHOD**
- **CTID**
- **SP-X** to **SP-W**

  You can enter safety positions for the axes. The control only approaches these positions if the machine manufacturer processes them in the NC macros.

Further information: "Pallet table", Page 2051

In the **Job list** workspace, you can activate or deactivate tool-oriented machining for each NC program via the context menu. This will also cause the control to update the **METHOD** column.

Further information: "Context menu", Page 1522

Sequence of tool-oriented machining

1 The entries TO and CTO tell the control that tool-oriented machining is effective for these rows of the pallet table
2 The control executes the NC program with the entry TO up to the TOOL CALL
3 The W-STATUS changes from BLANK to INCOMPLETE and the control enters a value into the CTID field
4 The control executes all other NC programs with the entry CTO up to the TOOL CALL
5 The control uses the next tool for the following machining steps if one of the following situations applies:
   - The next table row contains the entry PAL
   - The next table rowcontains the entry TO or WPO
   - There are rows in the table that do not yet contain the entry ENDED or EMPTY
6 The control updates the entry in the CTID field with each machining operation
7 If all table rows of the group contain the entry ENDED, the control processes the next rows of the pallet table

Block scan for mid-program startup

You can also return to a pallet table after an interruption. The control can show the rows and the NC block at which the interruption occurred.

The control saves the mid-program startup information in the **CTID** column of the pallet table.

The block scan in the pallet table is workpiece-oriented.

After a block scan, the control can resume tool-oriented machining if the tool-oriented machining method TO and CTO is defined in the subsequent rows.

Further information: "Pallet table", Page 2051

The following functions require special attention, particularly for mid-program startup:

- Changing the machine statuses with a miscellaneous function (e.g. M13)
- Writing to the configuration (e.g. WRITE KINEMATICS)
- Traverse range switchover
- Cycle **32**
- Cycle **800**
- Tilting the working plane

Notes

NOTICE
Danger of collision! Not all pallet tables and NC programs are suitable for tool-oriented machining. With tool-oriented machining, the control no longer executes the NC programs continuously, but divides them at the tool calls. The division of the NC programs allows functions that were not reset to be effective across programs (machine states). This leads to a danger of collision during machining! ▶ Consider the stated limitations ▶ Adapt pallet tables and NC programs to the tool-oriented machining ■ Reprogram the program information after each tool in every NC program (e.g. **M3** or **M4**). ■ Reset special functions and miscellaneous functions before each tool in every NC program (e.g., **Tilt the working plane** or **M138**) ▶ Carefully test the pallet table and associated NC programs in the **Program run, single block** operating mode

- If you want to start machining again, change the W-STATUS to BLANK or remove the previous input.

Notes on mid-program startup

- The entry in the CTID field remains there for two weeks. After this time, mid-program startup is no longer possible.
- Do not change or delete the entry in the CTID field.
- The data from the CTID field become invalid after a software update.
- The control saves the preset numbers for mid-program startup. If you change this preset, machining is shifted, too.
- Mid-program startup is no longer possible after editing an NC program within tool-oriented machining.

33.5 Pallet preset table

Application

Pallet presets are an easy way to compensate, for example, for mechanical differences between individual pallets.

The machine manufacturer defines the pallet preset table.

Related topics

- Contents of the pallet table

  Further information: "Pallet table", Page 2051
- Workpiece preset management

  Further information: "Preset management", Page 1025

Description of function

If a pallet preset is active, the workpiece preset is referenced to it.

In the **PALPRES** column of the pallet table, you can enter the corresponding pallet preset for a pallet.

You can also completely align the coordinate system to the pallet by, for example, positioning the pallet preset in the center of a clamping tower.

If a pallet preset is active, the control does not display an icon. You can check the active pallet preset and the defined values in the **Setup** application.

Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557

Note

NOTICE
Danger of collision! Despite a basic rotation based on the active pallet preset, the control does not display an icon in the status display. There is a risk of collision during all subsequent axis movements! ▶ Check the traverse movements of the machine ▶ Use pallet presets only in conjunction with pallets

If the pallet preset changes, you need to reset the workpiece preset.

Further information: "Setting a preset manually", Page 1028

34

Program Run

34.1 Program Run operating mode

34.1.1 Fundamentals

Application

In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either one block at a time or in full sequence.

You also execute pallet tables in this operating mode.

Related topics

- Executing individual NC blocks in the **MDI** application
  Further information: "The MDI Application ", Page 1933
- Creating NC programs
  Further information: "Programming fundamentals", Page 212
- Pallet tables
  Further information: "Pallet Machining and Job Lists", Page 1937

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- Copy the NC program and all called files to the **TNC:** drive

Description of function

> The following information also applies to pallet tables and job lists.

When you select a new NC program or when an NC program has been completely executed, the cursor is at the beginning of the program.

If you want to start machining at a different NC block, you first need to select the desired NC block by using the **Block scan** function.

Further information: "Block scan for mid-program startup", Page 1965

By default, the control runs NC programs in Full Sequence mode after the **NC Start** key has been pressed. In this mode, the control runs an NC program continuously up to its end, or up to a manual or programmed interruption.

In the **Single Block** mode, you execute each NC block separately by pressing the **NC Start** key.

The control shows the status of the machining process with the **Control-in-operation** icon in the status overview.

Further information: "Status overview on the TNC bar", Page 169

The **Program Run** operating mode provides the following workspaces:

- **GS** (option 44)

  Further information: "Global Program Settings (GPS, option 44)", Page 1217
- **Positions**

  Further information: "Positions workspace", Page 163
- **Program**

  Further information: "Program workspace", Page 217
- **Simulation**

  Further information: "Simulation Workspace", Page 1535
- **Status**

  Further information: "Status workspace", Page 171
- **Process Monitoring**

  Further information: "Process Monitoring workspace (option 168)", Page 1238

When you open a pallet table, the control will display the **Job list** workspace. You cannot modify this workspace.

Further information: "Job list workspace", Page 1938

Icons and buttons

The **Program Run** operating mode contains the following icons and buttons:

Icon or button	Meaning
	**Open file** With **Open file** you can open a file, for example an NC program. If you open a file, the control closes the file that was already open.
	Execution cursor The execution cursor shows which NC block is currently being run or is marked for running.
Single Block	If this toggle switch is active, then you run each NC block separately with the **NC Start** key. If Single Block mode is selected, then the operating mode's icon in the control bar changes.
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1366
Compensation tables	The control opens a selection menu with the following tables: ■ **D** ■ **T-CS** ■ **WPL-CS** **Further information:** "Compensation during program run", Page 1974
GOTO Cursor	The control marks the currently selected table row for execution. Active only if a pallet table is open (option 22) **Further information:** "Job list workspace", Page 1938
F limited	You activate or deactivate the feed-rate limitation for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 2096
AFC	You activate or deactivate Adaptive Feed Control (AFC, option 45). **Further information:** "AFC toggle switch in Program Run operating mode", Page 1201
AFC settings	The control opens a selection menu with the following tables for AFC (option 45): ■ **AFC.TAB** for AFC basic settings ■ **AFC.DEP** settings file for teach-in cuts of the active NC program ■ **AFC2.DEP** log file of the active NC program **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 1196
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC, option 145). **Further information:** "Active Chatter Control (ACC, option 145)", Page 1204
FMAX	You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit FMAX", Page 1958

Icon or button	Meaning
Breakpoints	When you select this button, the control opens the **Breakpoints** window with the following selection possibilities: ▪ **Feed FMAX** You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit FMAX", Page 1958 ▪ **Skip block** If this toggle switch is active, the control ignores NC blocks hidden with a **/** character. **Further information:** "Hiding NC blocks", Page 1513 If the toggle switch is active, the control grays out the NC blocks to be skipped. **Further information:** "Appearance of the NC program", Page 219 ▪ **Pause at M1** If this toggle switch is active, the control pauses the program run at each NC block with **M1**. **Further information:** "Overview of miscellaneous functions", Page 1319 If this toggle switch is inactive, the control grays out the **M1** syntax element. **Further information:** "Appearance of the NC program", Page 219
Skip block	If this toggle switch is active, the control ignores NC blocks hidden with a **/** character. **Further information:** "Hiding NC blocks", Page 1513 If the toggle switch is active, the control grays out the NC blocks to be skipped. **Further information:** "Appearance of the NC program", Page 219
Pause at M1	If this toggle switch is active, the control pauses the program run at each NC block with **M1**. **Further information:** "Overview of miscellaneous functions", Page 1319 If this toggle switch is inactive, the control grays out the **M1** syntax element. **Further information:** "Appearance of the NC program", Page 219
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** "GOTO function", Page 1511
Manual traverse	During a program run interruption you can move the axes manually. If **Manual traverse** is active, then the operating mode's icon in the control bar changes. **Further information:** "Manual traverse during an interruption", Page 1964
Edit	If this toggle switch is active, then you can edit the pallet table. Active only if a pallet table is open **Further information:** "Job list workspace", Page 1938
3D ROT	During a program run interruption you can move the axes manually in the tilted working plane (option 8). **Further information:** "Manual traverse during an interruption", Page 1964
Approach position	Return to contour after manual traverse of the machine axes during an interruption **Further information:** "Returning to the contour", Page 1972

Icon or button	Meaning
Block scan	The **Block scan** function allows you to start program run at any desired NC block. The control takes the preceding parts of the NC program up to this NC block into account mathematically; for example, whether the spindle was switched on with **M3**. **Further information:** "Block scan for mid-program startup", Page 1965
Open in the editor	The control opens the active NC program and also called NC programs in the **Editor** operating mode. Active only if an NC program is open **Further information:** "Editor operating mode", Page 216
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run.
Reset program	If you select **Internal stop**, the control activates this button. The control places the cursor back to the beginning of the program and resets any modally effective program information as well as the program run-time.

Feed rate limit FMAX

The **FMAX** button allows you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you have entered remains active across power cycles.

The **FMAX** button is available in the **MDI** application and in **Editor** operating mode.

When you select the **FMAX** button in the function bar, the control opens the **Feed rate FMAX** window.

If a feed rate limit is active, the control highlights the **FMAX** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the control shows the feed rate in orange.

Further information: "Statusanzeigen", Page

You deactivate the feed rate limit by entering a value of 0 in the **Feed rate FMAX** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt program run (e.g., with the miscellaneous function **M0**)
- Stop program run (e.g., with the **NC stop** key)
- Cancel the program run (e.g., with the **NC Stop** key in combination with the **INTERNAL STOP** button)
- Terminate program run (e.g., with the miscellaneous functions **M2** or **M30**)

Upon major errors, the control automatically aborts program run (e.g., during a cycle call with stationary spindle).

Further information: "Message menu on the information bar", Page 1532

If you run your NC program in **Single Block** mode or in the **MDI** application, the control will switch to the interrupted state after the execution of each NC block.

The control shows the current program run status with the **Control-in-operation** icon.

Further information: "Status overview on the TNC bar", Page 169

Below are some of the functions you can execute in an interrupted or canceled state:

- Selecting an operating mode
- Manual traverse of axes
- Checking Q parameters and changing these if necessary using the **Q INFO** function
- Changing the setting for the optional programmed interruption with **M1**
- Changing the setting for the programmed skipping of NC blocks with **/**

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the window **Q parameter list**
  - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

Programmed interruptions

You can set interruptions directly in the NC program. The control interrupts the program run in the NC block containing one of the following inputs:

- Programmed stop **STOP** (with and without miscellaneous function)
- Programmed stop **M0**
- Conditional stop **M1**

Resuming program run

After stopping the program with the **NC Stop** key or a programmed interruption, you can resume program run by pressing the **NC Start** key.

After canceling the program run with an **Internal stop**, you must start the program run at the beginning of the NC program or use the **Block scan** function.

After an interruption of the program run within a subprogram or program section repeat, you need to use the **Block scan** function for mid-program startup.

Further information: "Block scan for mid-program startup", Page 1965

Modally effective program information

The control saves the following data during a program interruption:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour (**Approach position** button).

Further information: "Returning to the contour", Page 1972

> The saved data remains active until it is reset (e.g., by selecting a program).

Notes

NOTICE
Danger of collision! Program cancellation, manual intervention, or forgotten resetting of NC functions or transformations can lead to the control performing unexpected or undesirable movements. This can lead to workpiece damage or collision. ▶ Rescind all programmed NC functions and transformations within the NC program ▶ Run a simulation before executing an NC program ▶ Check both the general as well as the additional status display for NC functions and transformations, such as an active basic rotation, before executing an NC program ▶ Carefully verify the NC program in the **Single Block** mode

- In the **Program Run** operating mode, the control marks active files with the status **M**, such as a selected NC program or tables. If you open such a file in another operating mode, the controls shows the status on the tab of the application bar.
- When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.
- You can adjust the feed rate and the spindle speed during program run with the potentiometers.
- If you modify the workpiece reference point during a program run interruption, you must re-select the NC block to resume.

  Further information: "Block scan for mid-program startup", Page 1965
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
- The settings in the **GS** workspace have an effect on program run, such as handwheel superimpositioning (option 44).

  Further information: "Global Program Settings (GPS, option 44)", Page 1217

Definitions

Abbreviation	Definition
GS (global program settings)	Global program settings
ACC (active chatter control)	Active Chatter Control

34.1.2 Navigation path in the Program workspace

Application

If you execute an NC program or a pallet table or if you test it in the opened **Simulation** workspace, the control displays a navigation path in the file information bar of the **Program** workspace.

The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when the program run is interrupted.

Related topics

- Program call

  Further information: "Selection functions", Page 388
- **Program** workspace

  Further information: "Program workspace", Page 217
- **Simulation** workspace

  Further information: "Simulation Workspace", Page 1535
- Interrupted program run

  Further information: "Interrupting, stopping or canceling program run", Page 1959

Requirement

- The **Program** and **Simulation** workspaces are both opened

  In the **Editor** operating mode you need both workspaces to use the function.

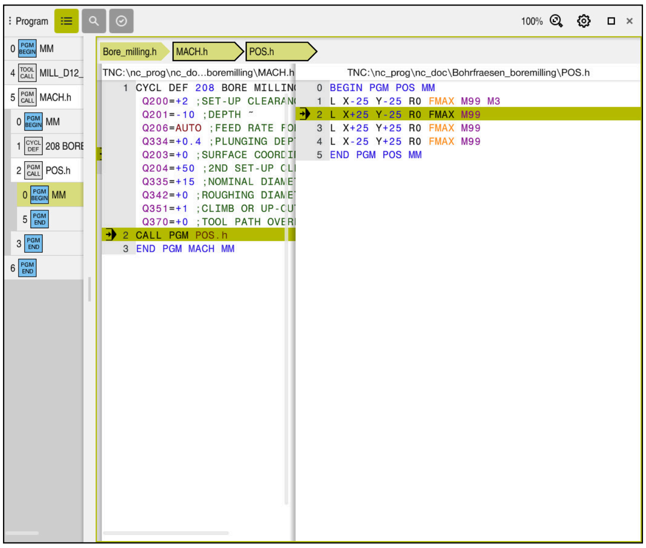
Description of function

The control shows the name of the NC program as a path element in the file information bar. As soon as the control calls a different NC program, the control adds a new path element with the name of the called NC program to the bar.

Additionally, the control displays the contents of the called NC program in a new layer in the **Program** workspace. The control displays as many NC programs side by side as the size of the workspace permits. If necessary, newly opened NC programs will cover previously opened NC programs. The control displays the covered NC programs in a narrow band at the left edge of the workspace.

When execution is interrupted, you can navigate between the NC programs. When you select the path element of an NC program, the control opens the content.

When you select the last path element, the control automatically marks the active NC block with the execution cursor. When you press the **NC Start** key, the control resumes execution of the NC program from this position.



NC programs called in the **Program** workspace in the **Program Run** operating mode

Depiction of path elements

The control displays the path elements of the navigation path as follows:

Format	Meaning
Black frame	The NC program is visible in the **Program** workspace and is not covered by other NC programs.
Highlighted in green	The NC program at the current cursor position is active or is considered for program run. If, for example, the cursor is positioned in the called NC program, the calling NC program will be considered for program run.
Highlighted in gray	The NC program is active for execution but will not be considered for program run at the current cursor position. If, for example, you stop the execution and navigate into the calling NC program, the control displays the path element of the called NC program in gray.

Note

In the **Program Run** operating mode the **Structure** column contains all structure items, including those of the called NC programs. The control indents the structure of the called NC programs.

The structure items allow you to navigate into every NC program. The control displays the associated NC programs in the **Program** workspace. The navigation path always remains at the position of the execution.

Further information: "Structure column in the Program workspace", Page 1514

34.1.3 Manual traverse during an interruption

Application

During a program run interruption you can move the machine axes manually.

Use the **Tilt the working plane (3D ROT)** window to assign the reference system for traversing the axes (option 8).

Related topics

- Manual traverse of machine axes

  Further information: "Moving the machine axes", Page 203
- Manual tilting of the working plane (option 8)

  Further information: "Tilting the working plane (option 8)", Page 1053

Description of function

When you select **Manual traverse**, you can move the axes with the axis keys of the control.

Further information: "Using axis keys to move the axes ", Page 204

In the **Tilt the working plane (3D ROT)** window, you can select the following functions:

Icon	Function	Meaning
	M-CS machine	Traversing in the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012
	**W-CS workpiece**	Traversing in the workpiece coordinate system **W-CS** **Further information:** "Workpiece coordinate system W-CS", Page 1016
	**WPL-CS working plane**	Traversing in the working plane coordinate system **WPL-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 1018
	**T-CS tool**	Traversing in the tool coordinate system **T-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 1018

When you select one of the functions, the control will display the associated icon in the **Positions** workspace. The control additionally shows the active coordinate system on the **3D ROT** button.

If **Manual traverse** is active, then the operating mode's icon in the control bar changes.

Notes

NOTICE

Danger of collision!

During a program interruption, you can move the axes manually (e.g., in order to retract from a hole when the working plane is tilted). There is a risk of collision if the **3-D ROT** setting is incorrect!

- It is better to use the **T-CS** function
- Use a low feed rate

- On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.

  Refer to your machine manual.

34.1.4 Block scan for mid-program startup

Application

With the **BLOCK SCAN** function you can start an NC program at any desired NC block. The control will arithmetically account for workpiece machining up to this NC block. For example, the control will switch on the spindle before the start.

Related topics

- Creating NC programs

  Further information: "Programming fundamentals", Page 212
- Pallet tables and job lists

  Further information: "Pallet Machining and Job Lists", Page 1937

Requirement

- The function must be enabled by your machine manufacturer.

  The **Block scan** function must be enabled and configured by your machine manufacturer.

Description of function

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- **Internal stop** button
- Emergency stop
- Power failure

If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute a block scan directly to the point of interruption. The control displays the message when you switch to **Program Run** operating mode for the first time.

You have the following options for a block scan:

- Block scan in the main program, with repetitions if necessary

  Further information: "Performing a single-level block scan", Page 1968
- Multi-level block scan in subprograms and touch probe cycles

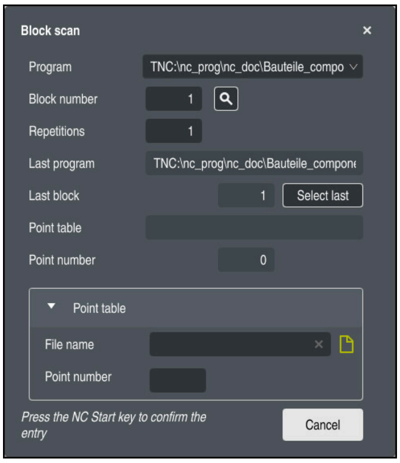
  Further information: "Performing a multi-level block scan", Page 1969
- Block scan in a point table

  Further information: "Block scan in point tables", Page 1970
- Block scan in pallet programs

  Further information: "Block scan in pallet tables", Page 1971

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During a block scan you can activate or deactivate **Single Block** mode.

Block scan window



Block scan window with the interruption point saved and the **Point table** area opened

The **Block scan** window provides the following data:

Row	Meaning
Pallet number	Row number in the pallet table
Program	Path of the active NC program
Block number	Number of the NC block at which program run should start Use the **search** icon to select the NC block in the NC program.
Repetitions	Number of the repetition for mid-program startup if the desired NC block is located within a program-section repeat.
Last pallet number	Pallet number that is active at the time of interruption Select the interruption point by using the **Select last** button.
Last program	Path of the NC program that is active at the time of interruption Select the interruption point by using the **Select last** button.
Last block	Number of the NC block that was active at the time of interruption Select the interruption point by using the **Select last** button.
Point file	Path of the point table In the **Point table** area
Point number	Row in the point table In the **Point table** area

Performing a single-level block scan

To start in an NC program by using a single-level block scan:

- Select the **Program Run** operating mode

Block scan

- Select **Block scan**
- The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed

Select last

- If required, use **Select last** to start at a saved interruption point
- Press the **NC Start** key
- The control starts the block scan and calculates up to the entered NC block.
- If you have changed the machine status, the control displays the **Restore machine status** window.
- Press the **NC Start** key
- The control restores the machine status (e.g., **TOOL CALL** or M functions).
- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1973

- Press the **NC Start** key
- The control resumes execution of the NC program.

Performing a multi-level block scan

If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level block scan. For this, you first go to the desired subprogram call and then continue the block scan. The same procedure is used for called NC programs.

To start in an NC program by using a multi-level block scan:

- Select the **Program Run** operating mode
- Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Perform a block scan to the first start-up point:

  Further information: "Performing a single-level block scan", Page 1968
- Activate the **Single Block** switch as needed
- Press the **NC Start** key to execute individual NC blocks as needed
- Select **Continue block scan**
- Define the NC block for mid-program startup
- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.
- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1973

- Select **Continue block scan** again as needed
- Repeat the steps
- Press the **NC Start** key
- > The control resumes execution of the NC program.

Block scan in point tables

To start in a point table:

- Select the **Program Run** operating mode
- Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Select **Point table**
- > The control opens the **Point table** area.
- **Point file**: Enter the path of the point table
- **Point number**: Select the row number of the point table for mid-program startup
- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.
- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1973

> If you would like to use the block scan function to start in a point pattern, then use the same procedure. Define the desired point for mid-program startup in the **Point number** field. The first point in the point pattern has the number 0.
>
> **Further information:** "Cycles for pattern definition", Page 431

Block scan in pallet tables

To start in a pallet table:

- Select the **Program Run** operating mode

Block scan

- Select **Block scan**
- > The control opens the **Block scan** window.
- **Pallet number**: Enter the row number of the pallet table
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed

Select last

- If required, use **Select last** to start at a saved interruption point
- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.
- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1973

> If the program run of a pallet table has been canceled, the control will suggest the most recently selected NC block of the most recently executed NC program as a point of interruption.

Notes

NOTICE

Danger of collision!

If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions, e.g. transformations. This means that there is a risk of collision during subsequent traversing movements!

- Use **GOTO** only when programming and testing NC programs
- Only use **Block scan** when executing NC programs

NOTICE

Danger of collision!

The **Block scan** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

- Use the **Block scan** function at multiple levels

- The control only displays the dialogs required by the process in the pop-up window.
- A **Block scan** always takes place in a workpiece-oriented manner, even if you selected a tool-oriented machining method. After the block scan, the control continues working again in accordance with the selected machining method.

  Further information: "Tool-oriented machining", Page 1948
- Even after an internal stop, the control shows the number of repetitions on the **LBL** tab of the **Status** workspace.

  Further information: "LBL tab", Page 175
- The **Block scan** function must not be used in conjunction with the following functions:
  - Touch probe cycles **0**, **1**, **3**, and **4** during the block scan search phase
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.

34.1.5 Returning to the contour

Application

With the **RESTORE POSITION** function, the control moves the tool to the workpiece contour in the following situations:

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the **INTERNAL STOP** function.
- Return to the contour after a block scan (e.g., after an interruption with **INTERNAL STOP**)
- Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption

Related topics

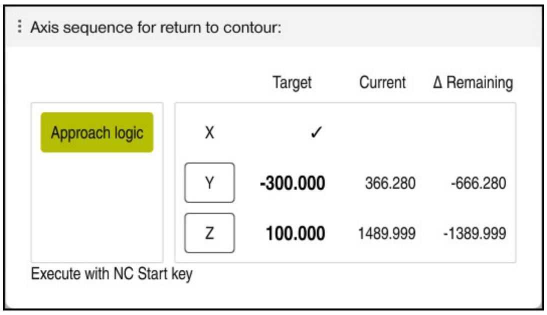
- Manual traverse during program run interruptions
  Further information: "Manual traverse during an interruption", Page 1964
- **Block scan** function
  Further information: "Block scan for mid-program startup", Page 1965

Description of function

If you have selected the **Manual traverse** button, this button will change to **Approach position**.

When you select **Approach position**, the control will open the **Axis sequence for return to contour:** window.

Axis sequence for return to contour: window



Axis sequence for return to contour: window

In the **Axis sequence for return to contour:** window, the control displays all of the axes that are not yet located at the correct position for program execution.

The control suggests a positioning logic for the sequence of the traversing movements. If the tool is located in the tool axis below the position to be approached, then the control offers the tool axis as the first traverse direction. You can also traverse the axes in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 1973

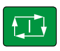
If manual axes are included in the axes to be returned to the contour, then the control will not suggest a positioning logic. As soon as you have correctly positioned the manual axis, the control will suggest a positioning logic for the remaining axes.

Further information: "Positioning manual axes", Page 1974

Positioning the axes in a self-selected sequence

To position the axes in a self-selected sequence:

- ▶ Select **Approach position**
- > The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- ▶ Select the desired axis (e.g., **X**)



- ▶ Press the **NC Start** key
- > The control moves the axis to the required position.
- > When the axis has reached the correct position, the control shows a checkmark under **Target**.
- ▶ Position the remaining axes
- > When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:

Approach position

- ▶ Select **Approach position**
- > The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- ▶ Select the manual axis (e.g., **W**)
- ▶ Position the manual axis to the value shown in the window
- > When a manual axis with encoder has reached the position, the control automatically clears the value.
- ▶ Select **Axis in position**
- > The control saves the position.

Note

In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

Definition

Manual axis
Manual axes are non-driven axes that need to be positioned by the machine operator.

34.2 Compensation during program run

Application

During program run, you can open the selected compensation tables and the active datum table, and edit the values.

Related topics

- Using compensation tables

  Further information: "Tool compensation with compensation tables", Page 1120
- Editing compensation tables in the NC program

  Further information: "Accessing table values ", Page 1991
- Contents and creation of compensation tables

  Further information: "Compensation table *.tco", Page 2055

  Further information: "Compensation table *.wco", Page 2057
- Contents and creation of a datum table

  Further information: "Datum table", Page 1033
- Activating a datum table in the NC program

  Further information: "Datum table", Page 2045

Description of function

The control opens the selected tables in **Tables** operating mode.

The changed values do not take effect until the compensation or the datum has been activated again.

34.2.1 Opening tables from within Program Run operating mode

To open the compensation tables from within **Program Run** operating mode:

Compensation tables

- Select **Compensation tables**
- > The control displays a selection menu.
- Select the desired table
  - **D**: Datum table
  - **T-CS**: Compensation table ***.tco**
  - **WPL-CS**: Compensation table ***.wco**
- > The control opens the selected table in **Tables** operating mode.

Notes

NOTICE

Danger of collision!

The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.

- Make sure to confirm any changes made to the table immediately (e.g., by pressing the **ENT** key)
- Activate the datum or compensation value in the NC program again
- Carefully test the NC program after changing the table values

- When you open a table in **Program Run** operating mode, the control will display the status **M** on the tab of the table. This status indicates that this table is active for program run.
- The clipboard allows you to transfer axis positions from the position display to the datum table.

  Further information: "Status overview on the TNC bar", Page 169

34.3 Retract application

Application

The **Retract** application allows you to disengage the tool from the workpiece after an interruption in power (e.g., retraction of a tap engaged in the workpiece).

You can also retract a tool when the working plane is tilted or retract an inclined tool.

Requirement

- This application must be enabled by your machine manufacturer.

  The machine parameter **retractionMode** (no. 124101) allows the machine manufacturer to define whether the control will display the **Retract** toggle switch during start-up.

Description of function

The **Retract** application provides the following workspaces:

- **Retract**

  Further information: "Retract workspace", Page 1977
- **Positions**

  Further information: "Positions workspace", Page 163
- **Status**

  Further information: "Status workspace", Page 171

The **Retract** application provides the following buttons in the function bar:

Button	Meaning
Retract	Retract the tool with the axis keys or the electronic handwheel
End retraction	Close the **Retract** application The control opens the **End retraction?** window and prompts you to answer a confirmation request.
Start values	Reset the entries in the **A**, **B**, **C**, and **Thread pitch** fields to their original values

You select the **Retract** application by using the **Retract** toggle switch if the following conditions apply during start-up:

- Power interrupted
- No control voltage for the relay
- **Move to ref. point** application

If you have activated a feed rate limit before the power failure occurred, this feed rate limit will still be active. When you select the **Retract** button, the control will display a pop-up window: This window allows you to deactivate the feed rate limit.

Further information: "Feed rate limit FMAX", Page 1958

Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction: ■ **Machine axes**: Traverse in the machine coordinate system **M-CS** ■ **Tilted system**: Traverse in the working plane coordinate system **WPL-CS** (option 8) ■ **Tool axis**: Traverse in the tool coordinate system **T-CS** (option 8) ■ **Thread**: Traverse in the tool coordinate system **T-CS** with compensating movements of the spindle **Further information:** "Reference systems", Page 1010
Kinematics	Name of the active machine kinematics
A, **B**, **C**	Current position of the rotary axes Effective for **Tilted system** traverse mode
Thread pitch	Thread pitch from the **PITCH** column of tool management Effective for **Thread** traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool: ■ **Right-hand thread** ■ **Left-hand thread** Effective for **Thread** traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect Effective for **Tool axis** traverse mode

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are able to reset them manually.

Note

NOTICE

Caution: Danger to the tool and workpiece!

A power failure during the machining operation can cause uncontrolled “coasting” or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

- Use a low feed rate
- Please keep in mind that the traverse range monitoring is not available for non-referenced axes

Example

The power failed while a thread cutting cycle in the tilted working plane was being performed. You have to retract the tap:

- Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace
- Activate the **Retract** toggle switch
- Press **OK**
- > The control compiles the PLC program.
- Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit
- > The control opens the **Retract** application and displays the **Assume position values?** window.
- Compare the displayed position values with the actual position values
- Select **OK**
- > The control closes the **Assume position values?** window
- Select the **Thread** traverse mode as needed
- Enter the thread pitch as needed
- Enter the direction of rotation as needed
- Select **Retract**
- Retract the tool with the axis keys or the handwheel
- Select **End retraction**
- > The control opens the **End retraction?** window and prompts you to answer a confirmation request.
- If the tool was correctly retracted, select **Yes**
- > The control closes the **End retraction?** window and the **Retract** application.

35

Tables

35.1 Tables operating mode

Application

In the **Tables** operating mode you can open various tables and edit them as necessary.

Description of function

If you select **Add**, the control displays the **Quick selection** and **Open File** workspaces.

In the **Quick selection** workspace, you can open some tables directly.

Further information: "Quick selection workspace", Page 1154

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: "Open File workspace", Page 1153

Multiple tables can be open at the same time. The control displays each table in a separate workspace.

If a table is selected for program run or simulation, the control shows the status **M** or **S** on the tab of the application. The status of the active application is highlighted in color and for the remaining applications in gray.

You can open the **Table** and **Form** workspaces in every application.

Further information: "Table workspace", Page 1982

Further information: "Form workspace for tables", Page 1989

You can select various functions by using the context menu (e.g., **Copy**).

Further information: "Context menu", Page 1522

Buttons

The **Tables** operating mode provides the following buttons in the function bar:

Button	Meaning
Activate the preset	The control activates the currently selected row of the preset table as preset. **Further information:** "Preset table", Page 2035
Undo	The control undoes the last change.
Redo	The control restores the change that has been undone.
GOTO record	The control opens the **GOTO jump instruction** window. The control jumps to the row number you have defined.
Edit	If the toggle switch is active, you can edit the table.
Insert tool	The control opens the **Insert tool** window that allows you to add a new tool to tool management. **Further information:** "Tool management ", Page 297 When you select the **Append** check box, the control inserts the tool below the last row of the table.
Insert line	The control inserts a row at the end of the table.
Reset row	The control resets all data contained in the row.
Delete tool	The control deletes the tool selected in the tool management. **Further information:** "Tool management ", Page 297
Delete row	The control deletes the currently selected row.
Lock record	The control locks the currently selected row of the preset table and thus protects the contents from changes. **Further information:** "Write-protection for table rows", Page 2040
Mark row	The control marks the currently selected row.
Import	The control imports tool data. **Further information:** "Importing tool data", Page 299
Inspect	The control inspects a tool.
Unload	The control unloads a tool.
Load	The controls loads a tool.

> Refer to your machine manual.
> If necessary, the machine manufacturer adapts the buttons.

35.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell

- Activate **Edit**
- > The control enables the values for editing.

> If the **Edit** toggle switch is active, you can edit the contents in both the **Table** workspace and the **Form** workspace.

Notes

- The control enables you to transfer tables from previous controls to the TNC7 and to adapt them automatically, if needed.
- If you open a table which has columns missing, the control will open the the **Incomplete table layout** window.

  In the **Incomplete table layout** window a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.
- If you, for example, have processed tables in a text editor, the control offers the **Update TAB / PGM** function. Use this function to complete an incorrect table format.

  Further information: "File management", Page 1144

> Edit tables only by using the table editor in the **Tables** operating mode to avoid errors (e.g., in the format).

35.2 Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.

Description of function

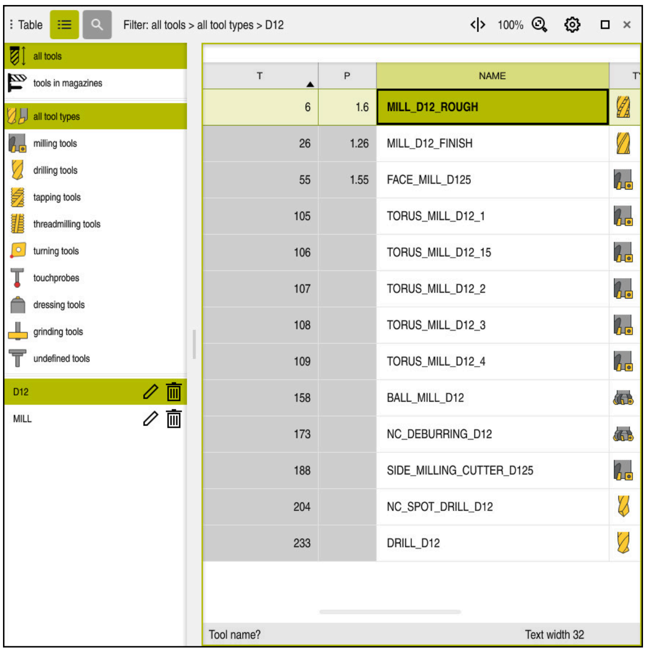


Table workspace

In **Tables** operating mode, the **Table** workspace is open in every application by default.

The control displays the name and path of the file above the header of the table.

When you select the title of a column, the control will sort the table contents by this column.

If the table allows it, you can also edit the table contents in this workspace.

Icons and shortcuts

The **Table** workspace provides the following icons or shortcuts:

Icon or shortcut	Function
	Open the filters **Further information:** "Filter column in the Table workspace", Page 1983
	Open the search function **Further information:** "Search column in the Table workspace", Page 1986
	Change column width **Further information:** "Changing the column width in the Table workspace", Page 1988
100%	Font size of the table If you select the percent value, the control displays symbols for increasing and decreasing the font size.
	Set the font size of the table to 100%
	Open the settings in the **Tables** window **Further information:** "Settings in the Table workspace", Page 1986
CTRL+A	Mark all rows
CTRL+BLANK	Mark the active row or end the marking function
SHIFT+↑	Additionally mark the row above
SHIFT+↓	Additionally mark the row below

Filter column in the Table workspace

You can filter the following table types:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

Filtering in Tool management

The control provides the following default filters in the **Tool management**:

- **All tools**
- **Magazine tools**

According to the selection of **All tools** or **Magazine tools**, the control additionally offers the following default filters in the filter column:

- **All types**
- **Milling cutters**
- **Drills**
- **Taps**
- **Thread cutters**
- **Lathe tools**
- **Touch probes**
- **Dressing tools**
- **Grinding tools**
- **Undefined tools**

To display specific tool types, you must activate the desired filter or filters and deactivate the **All types** filter.

Filters in the Pocket table

The control provides the following default filters in the **Pocket table**:

- **all pockets**
- **spindle**
- **main magazine**
- **empty pockets**
- **occupied pockets**

Filtering in the Presets tablePresets

The control provides the following default filters in the **Presets** table:

- **Base transformations**
- **Offsets**
- **SHOW ALL**

User-defined filters

You can additionally create user-defined filters.

The control provides the following icons for each user-defined filter:

Icon	Meaning
✎	If you click on **Editing**, the control opens the **Search** column. You can edit and save the selected filter or save a filter under a new name. **Further information:** "Search column in the Table workspace", Page 1986
🗑	You can delete the selected filter.

To deactivate the user-defined filter, you need to activate the **All** filter and then to deactivate the user-defined filter.

> Refer to your machine manual.
> This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions for the machine.

Logical connective operations between requirements and filters

The control connects the filters as follows:

- AND operation for several requirements within one filter

  You create, for example, a user-defined filter that contains the requirements **R = 8** and **L > 150**. The control filters the table rows when you activate this filter. The control displays only the table rows that meet both requirements at the same time.
- OR operation between filters of the same type

  When you activate the default filters **Milling cutters** and **Lathe tools**, for example, the control filters the table rows. The control displays only the table rows that meet at least one of the requirements. The table row must contain either a milling cutter or a turning tool.
- AND operation between filters of different types

  You create, for example, a user-defined filter that contains the requirement **R > 8**. When you activate this filter and the default filter **Milling cutters**, the control filters the table rows. The control displays only the table rows that meet both requirements at the same time.

Search column in the Table workspace

You can search the following table types:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

You can define multiple search conditions in the search function.

Each condition includes the following information:

- Table column, such as **T** or **NAME**
  Use the **Search in** selection menu to select the column.
- Perhaps an operator, such as **Contains** or **Equal to (=)**
  Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field

> If you search the columns using predefined selection values, the control offers a selection menu instead of the input field.

The control provides the following buttons:

Button	Meaning
+	Use **Add** to add several conditions. The conditions will have a combined effect when you perform the search. You can save several conditions in a user-defined filter.
Search	The control searches the table.
Reset	The control resets the entered conditions and removes any additional conditions.
Save	You can save the entered conditions as a filter. You can assign any name to the filter.

> Refer to your machine manual.
> This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions for the machine.

Settings in the Table workspace

In the **Tables** window, you can influence the shown contents of the **Table** workspace.

The **Tables** window consists of the following areas:

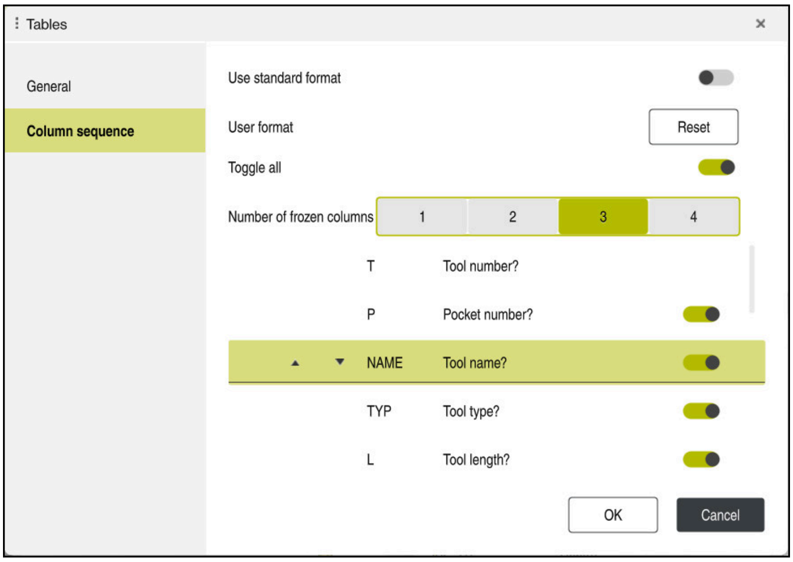
- **General**
- **Column sequence**

General area

The setting selected in the **General** area is modally effective.

If the **Synchronize table and form** toggle switch is active, the cursor will move synchronously. If, for example, you select a different table column in the **Table** workspace, the control moves the cursor synchronously in the **Form** workspace.

Column sequence area



Tables window

The **Column sequence** area contains the following settings:

Setting	Meaning
Use standard format	If you activate the toggle switch, the control shows all table columns, indicating them in the standard sequence. If you deactivate the toggle switch, the control restores the previous setting.
User format	If you select the **Reset** button, the control resets the adaptations to the settings of the standard format.
Toggle all	If you activate the toggle switch, the control shows all table columns. If you deactivate the toggle switch, the control hides all table columns. The first column in each table cannot be hidden.
Number of frozen columns	You define how many table columns the control freezes at the left table edge. You can freeze up to four table columns. These table columns will remain visible even when you navigate further to the right within the table.
Columns of the currently opened table	The control displays all table columns below each other. Use the toggle switches to separately hide or show each table column. The control displays a line below the selected number of frozen columns. When you select a table column, the control displays up and down arrows. Use these arrows to change the sequence of the columns. The respective first column in the table cannot be shifted.

The settings in the **Column sequence** area only apply to the currently opened table.

35.2.1 Changing the column width in the Table workspace

To change the column width:

- Select the table column
- Select **Change column width**
- The control displays an arrow on the left and right in the header of the selected table column.
- Drag the arrow to the left or right
- The control reduces or enlarges the table column.
- Select other table columns if necessary

> If you select a further table column, then you need to select **Change column width** again.

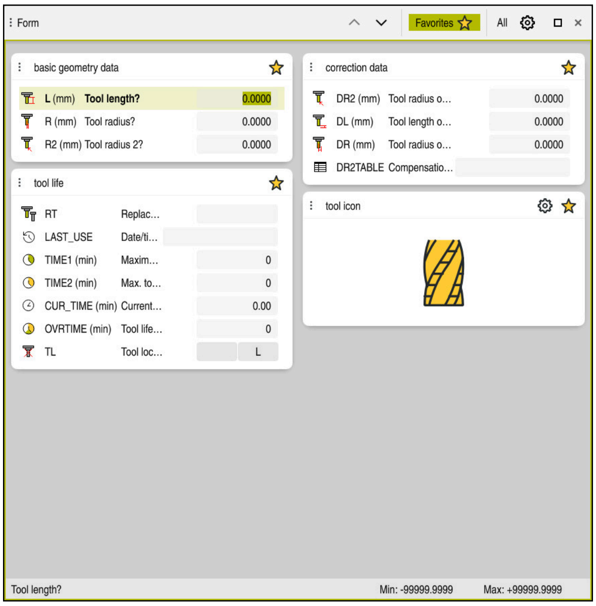
> You can also change the column width of non-editable table columns.

35.3 Form workspace for tables

Application

In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.

Description of function



Form workspace in the **Favorites** view

The control displays the following information for each column:

- Icon of the column as needed
- Name of the column
- Unit of measure as needed
- Column description
- Current value

The control displays an icon of the selected tool type in the **Tool Icon** area. For the turning tools the icons also take into account the tool orientation and show where the relevant tool data will apply.

Further information: "Tool types", Page 281

If an input is invalid, the control displays an icon ahead of the input field. When you tap this icon, the control shows the cause of the error (e.g., **Too many characters**).

The control displays the contents of specific tables in groups within the **Form** workspace. In the **All** view, the control shows all groups. Use the **Favorites** function to select individual groups in order to configure a customized view. Use the gripper to arrange the groups.

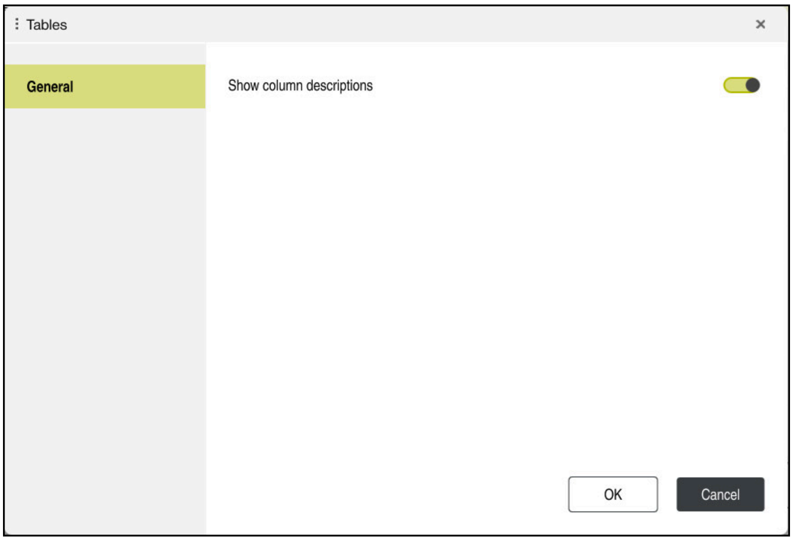
Icons

The **Table** workspace provides the following icons:

Icon or shortcut	Function
  **SHIFT+↑** **SHIFT+↓**	Navigate between table rows
	■ Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 1990 ■ Change the size of the graphic in the **Tool Icon** area The control opens a selection window with the following settings: ■ **Small** ■ **Medium** ■ **Large**
	Favorite

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the column descriptions. The selected setting is modally effective.



35.4 Accessing table values

35.4.1 Fundamentals

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.

The control provides the following functions for accessing the table values:

Syntax	Function	Further information
TABDATA READ	Read the value from a table cell	Page 1992
TABDATA WRITE	Write a value to a table cell	Page 1993
TABDATA ADD	Add a value to a table value	Page 1994

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.

Related topics

- Fundamentals regarding variables

  Further information: "Basics", Page 1362
- Tool table

  Further information: "Tool table tool.t", Page 1995
- Compensation tables

  Further information: "Compensation tables", Page 2055
- Reading values from freely definable tables

  Further information: "Reading a freely definable table with FN 28: TABREAD", Page 1395
- Writing values to freely definably tables

  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 1394

35.4.2 Reading table values with TABDATA READ

Application

The function **TABDATA READ** allows you to read a value from a table and save it to a Q parameter.

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.

Description of function

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.

Input

11 TABDATA READ Q1 = CORR-TCS COLUMN "DR" KEY "5"	; Save the value in row 5, column **DR**, from the compensation table to **Q1**

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
READ	Reading a table value
Q/QL/QR or **QS**	Type of variable and number in which the control saves the value
TOOL, **CORR-TCS**, **CORR-WPL** or **PRESET**	Read the value from the tool table or a compensation table ***.tco** or ***.wco** or from the preset table
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name

35.4.3 Writing table values with TABDATA WRITE

Application

The function **TABDATA WRITE** allows you to write a value from a Q parameter into a 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.

Description of function

Depending on the type of column you want to write to, you can use **Q**, **QL**, **QR**, or **QS** as a transfer parameter.

Input

11 TABDATA WRITE CORR-TCS COLUMN "DR" KEY "3" = Q1	; Write the value from **Q1** to row 5, column **DR**, of the compensation table

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
WRITE	Writing a table value
CORR-TCS, **CORR-WPL** or **PRESET**	Write a value to a compensation table ***.tco** or ***.wco** or to the preset table
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name
Q/QL/QR or **QS**	Type of variable and number that contains the value to be written

35.4.4 Adding table values with TABDATA ADD

Application

The function **TABDATA ADD** allows you to add a value from a Q parameter to a value contained in the table.

You can use the **TABDATA ADD** function to update a tool compensation value after a measurement has been repeated, for example.

Description of function

Depending on the type of column you want to write to, you can use **Q**, **QL**, or **QR** as a transfer parameter.

In order to write into a compensation table, you need to activate the table.

Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 1122

Input

11 TABDATA ADD CORR-TCS COLUMN "DR" KEY "3" = Q1	; Add the value from **Q1** to row 5, column **DR**, of the compensation table

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
ADD	Adding a value to a table value
CORR-TCS, **CORR-WPL** or **PRESET**	Write a value to a compensation table ***.tco** or ***.wco** or to the preset table
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name
Q/QL/QR	Type of variable and number that contains the value to be added

35.5 Tool tables

35.5.1 Overview

This chapter describes the tool tables of the control.

- Tool table **tool.t**

  Further information: "Tool table tool.t", Page 1995
- Turning tool table **toolturn.trn** (option 50)

  Further information: "Turning tool table toolturn.trn (option 50)", Page 2006
- Grinding tool table **toolgrind.grd** (option 156)

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010
- Dressing tool table **tooldress.drs** (option 156)

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019
- Touch probe table **tchprobe.tp**

  Further information: "Touch probe table tchprobe.tp", Page 2022

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 297

35.5.2 Tool table tool.t

Application

The tool table **tool.t** contains the data specific to drilling and milling tools. The tool table also contains all tool data that are independent of the technology, such as the tool life **CUR_TIME**.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 297
- Tool data required for milling or drilling tools

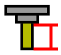
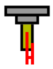
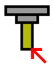
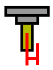
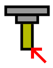
  Further information: "Tool data for milling and drilling tools", Page 285

Description of function

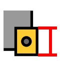
The file name of the tool table is **tool.t** and the tool table must be stored in the folder **TNC:\table**.

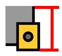
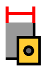
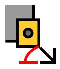
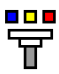
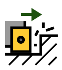
The **tool.t** tool table provides the following parameters:

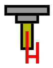
Parameter	Meaning
T	**Tool number?** Row number in the tool table The tool number allows identifying each tool unambiguously (e.g., for calling a tool). **Further information:** "Tool call by TOOL CALL", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 This parameter applies to all tools, regardless of technology. Input: **0.0...32767.9**

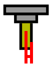
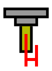
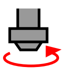
Parameter	Meaning
NAME	**Tool name?** The tool name identifies a tool, for example when calling it. **Further information:** "Tool call by TOOL CALL", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 This parameter applies to all tools, regardless of technology. Input: **Text width 32**
L 	**Tool length?** Length of tool, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**
R 	**Tool radius?** Tool radius, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**
R2 	**Tool radius 2?** Corner radius for the exact definition of the tool for three-dimensional radius compensation, graphic representation and collision monitoring of, for example, ball-nose cutters or toroid cutters. **Further information:** "3D tool compensation (option 9)", Page 1126 Input: **-99999.9999...+99999.9999**
DL 	**Tool length oversize?** Delta value of tool length as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to the parameter **L** Input: **-999.9999...+999.9999**
DR 	**Tool radius oversize?** Delta value of tool radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to parameter **R** Input: **-999.9999...+999.9999**
DR2 	**Tool radius oversize 2?** Delta value of tool radius 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to parameter **R2** Input: **-999.9999...+999.9999**

Parameter	Meaning
TL 	**Tool locked?** Tool is enabled or locked for machining: ■ No value entered: Enabled ■ **L**: Locked The control locks the tool after exceeding maximum tool age **TIME1**, maximum tool age 2 **TIME2** or after exceeding one of the parameters for automatic tool measurement. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: No value, **L**
RT	**Replacement tool?** Number of the replacement tool If the control calls a tool in a TOOL CALL and the tool is not available or locked, the control inserts the replacement tool. If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool at a suitable location. **Further information:** "Automatically inserting a replacement tool with M101", Page 1355 If the replacement tool is not available or locked, the control inserts the replacement tool of the replacement tool. You can define an index after the period. **Further information:** "Indexed tool", Page 276 If you define the value 0, the control will not use a replacement tool. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **0.0...32767.9**
TIME1 	**Maximum tool age?** Maximum tool age in minutes If the current tool age **CUR_TIME** exceeds the **TIME1** value, the control locks the tool and displays an error message when the tool is called the next time. The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99999**

Parameter	Meaning
TIME2 	**Max. tool age for TOOL CALL?** Maximum tool age 2 in minutes The control inserts a replacement tool in the cases below: ■ When the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool. The control no longer inserts the tool when the tool is called. If a replacement tool **RT** is defined and available in the magazine, the control inserts the replacement tool. If no replacement tool is available, the control will display an error message. ■ If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool **RT** at a suitable location. **Further information:** "Automatically inserting a replacement tool with M101", Page 1355 The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99999**
CUR_TIME 	**Current tool age?** The current tool age equals the time during which the tool is cutting a workpiece. The control counts this time automatically and enters the current tool age in minutes. You can edit the tool age of an active tool during program run after you have inserted an indexable insert, for example. The control will directly apply the value to tool life monitoring. This parameter applies to all tools, regardless of technology. Input: **0...99999.99**
TYPE	**Tool type?** Depending on the selected tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Tool types", Page 281 **Further information:** "Tool management ", Page 297 This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **MILL**, **MILL_R**, **MILL_F**, **MILL_FACE**, **BALL**, **TORUS**, **MILL_CHAMFER**, **DRILL**, **TAP**, **CENT**, **TURN**, **TCHP**, **REAM**, **CSINK**, **TSINK BOR**, **BCKBOR**, **GF**, **GSF**, **EP**, **WSP**, **BGF**, **ZBGF**, **GRIND** and **DRESS**
DOC	**Tool description** This parameter applies to all tools, regardless of technology. Input: **Text width 32**
PLC	**PLC status?** Tool information for the PLC Refer to your machine manual. This parameter applies to all tools, regardless of technology. Entry: **%00000000...%11111111**
LCUTS 	**Tooth length in the tool axis?** Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring. Input: **-99999.9999...+99999.9999**

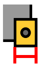
Parameter	Meaning
LU 	**Usable length of the tool?** Usable length of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of, for example, necks of end mills. Input: **0.0000...999.9999**
RN 	**Neck radius of the tool?** Neck radius for the exact definition of the tool for graphic simulation and collision monitoring of e.g. neck of end mills or side milling cutters. Only if the useful length **LU** is longer than the **LCUTS** length of cutting edge, can the tool contain a neck radius **RN**. Input: **0.0000...999.9999**
ANGLE 	**Maximum plunge angle?** Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles. Input: **-360.00...+360.00**
CUT 	**Number of teeth?** Number of teeth of the tool for automatic tool measurement or cutting data calculation. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 **Further information:** "Cutting data calculator", Page 1529 This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...99**
TMAT 	**Tool material?** Tool material from the tool material table **TMAT.tab** for cutting data calculation. **Further information:** "Table for tool materials TMAT.tab", Page 2048 Selection by means of a selection window Input: **Text width 32**
CUTDATA 	**Cutting data table?** **Further information:** "Cutting data calculator", Page 1529 Select cutting data table with the ***.cut** or ***.cutd** file extensions for cutting data calculation. **Further information:** "Cutting data table *.cut", Page 2049 Selection by means of a selection window Entry: **Text width 20**

Parameter	Meaning
LTOL 	**Wear tolerance: length?** Permitted tool length deviation in wear detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ▪ Milling and drilling tools ▪ Turning tools (option 50) Input: **0.0000...5.0000**
RTOL 	**Wear tolerance: radius?** Permitted tool radius deviation in wear detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ▪ Milling and drilling tools ▪ Turning tools (option 50) Input: **0.0000...5.0000**
R2TOL	**Wear tolerance: Radius 2?** Permitted tool radius 2 deviation in wear detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ▪ Milling and drilling tools ▪ Turning tools (option 50) Input: **0...9.9999**
DIRECT 	**Cutting direction?** Cutting direction of the tool for automatic tool measurement with a rotating tool: ▪ **-**: **M3** ▪ **+**: **M4** **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 This parameter applies to the following tools, regardless of technology: ▪ Milling and drilling tools ▪ Turning tools (option 50) Input: **-**, **+**

Parameter	Meaning
R-OFFS 	**Tool offset: radius?** Position of tool upon length measurement, offset between the center of the tool touch probe and the tool center for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **-99999.9999...+99999.9999**
L-OFFS 	**Tool offset: length?** Position of tool upon radius measurement, distance between the top edge of the tool touch probe and the tool tip for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 Is added to the machine parameter **offsetToolAxis** (no. 122707) This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **-99999.9999...+99999.9999**
LBREAK 	**Breakage tolerance: length?** Permitted tool length deviation in breakage detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0.0000...9.0000**
RBREAK 	**Breakage tolerance: radius?** Permitted tool radius deviation in breakage detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1906 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0.0000...9.0000**
NMAX 	**Maximum speed [rpm]** Limitation of spindle speed for the programmed value including control by the potentiometer. Input: **0...999999**

Parameter	Meaning
LIFTOFF	**Lift-off allowed?** Automatic tool lift-off with active **M148** or allow **FUNCTION LIFTOFF**: ■ **Y**: activate **LIFTOFF** ■ **N**: deactivate **LIFTOFF** **Further information:** "Automatically lifting off upon an NC stop or a power failure with M148", Page 1352 **Further information:** "Automatic tool liftoff with FUNCTION LIFTOFF", Page 1191 Selection by means of a selection window Input: **Y**, **N**
TP_NO	**Number of the touch probe** Number of touch probe in touch probe table **tchprobe.tp** **Further information:** "Touch probe table tchprobe.tp", Page 2022 Input: **0...99**
T-ANGLE 	**Point angle** Point angle of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of drills, for example. **Further information:** "Cycles for milling", Page 507 Input: **-180...+180**
LAST_USE	**Date/time of last tool usage** Time of last tool presence in the spindle This parameter applies to all tools, regardless of technology. Input: **00:00:00 01.01.1971**...**23:59:59 31.12.2030**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table **Further information:** "Pocket table tool_p.tch", Page 2026 Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
AFC	**Feedback-control strategy** Control setting for adaptive feed control AFC (option 45) from the **AFC.tab** table **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 1196 Selection by means of a selection window Entry: **Text width 10**
ACC	**ACC active?** Activate or deactivate active chatter control ACC (option 145): ■ **Y**: activate ■ **N**: deactivate **Further information:** "Active Chatter Control (ACC, option 145)", Page 1204 Selection by means of a selection window Input: **Y**, **N**

Parameter	Meaning
PITCH 	**Tool thread pitch?** Thread pitch of the tool for automatic calculations within cycles. A positive sign means a right-hand thread. **Further information:** "Cycles for milling", Page 507 Input: **-9.9999...+9.9999**
AFC-LOAD	**Reference power for AFC [%]** Tool-dependent reference power for AFC (option 45). The input in percent refers to the rated spindle power. The control immediately uses the value given for feedback control, meaning a teach-in cut is dropped. Calculate the value beforehand with a teach-in step. **Further information:** "AFC teach-in cut", Page 1202 Input: **1.0...100.0**
AFC-OVLD1	**AFC overload warning level [%]** Cut-related tool wear monitoring for AFC (option 45). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. **Further information:** "Monitoring tool wear and tool load", Page 1203 Input: **0.0...100.0**
AFC-OVL2	**AFC ovrload switch-off level [%]** Cut-related tool load monitoring for AFC (option 45). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. **Further information:** "Monitoring tool wear and tool load", Page 1203 Input: **0.0...100.0**
KINEMATIC	**Tool-carrier kinematics** Assigning a tool carrier for exact definition of the tool for graphical simulation and collision monitoring. **Further information:** "Tool carrier management", Page 301 Selection by means of a selection window This parameter applies to all tools, regardless of technology. Entry: **Text width 20**
DR2TABLE	**Compensation val. table for DR2** Assigning a compensation value table ***.3dtc** for 3D tool radius compensation depending on the contact angle (option 92). This allows the control to compensate for inaccurate shapes of a ball-nose cutter or the deflection behavior of a touch probe, for example. **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1140 Selection by means of a selection window Entry: **Text width 16**

Parameter	Meaning
OVRTIME 	**Tool life expired** Time in minutes during which the tool may be used beyond the tool life defined in column **TIME2**. The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control uses the parameter when searching for tool names. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
RCUTS 	**Width of the indexable insert** Face-side width of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., for indexable inserts). Input: **0...99999.9999**
DB_ID	**ID for central tool management** The database ID allows you to identify a tool (e.g., by using client applications within a tool management system). **Further information:** "Database ID", Page 276 For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool. **Further information:** "Indexed tool", Page 276 This parameter applies to all tools, regardless of technology. Input: **Text width 40**
R_TIP	**Radius at the tip** Radius at the tool tip for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of countersinks, for example. Input: **0.0000...999.9999**

Notes

- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 2026
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
  - 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 least remaining tool life.
- In the machine parameter **offsetToolAxis** (no. 122707), the machine manufacturer defines the distance between the upper edge of the tool touch probe and the tool tip.

  The parameter **L-OFFS** is added to this defined distance.
- In the machine parameter **zeroCutToolMeasure** (no. 122724) the machine manufacturer defines whether the control takes the parameter **R-OFFS** into account for automatic tool measurement.

35.5.3 Turning tool table toolturn.trn (option 50)

Application

The turning tool table **toolturn.trn** contains the data specific to turning tools.

Related topics

- Editing tool data in tool management
  Further information: "Tool management ", Page 297
- Tool data required for turning tools
  Further information: "Tool data for turning tools (option 50)", Page 287
- Milling-turning operations on the control
  Further information: "Turning (option 50)", Page 236
- General tool data, regardless of the technology
  Further information: "Tool table tool.t", Page 1995

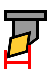
Requirements

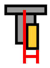
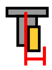
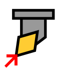
- Combined milling-turning (software option 50)
- Turning tool is defined in **TYP** column of tool management
  Further information: "Tool types", Page 281

Description of function

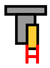
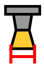
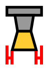
The file name of the turning tool table is **toolturn.trn** and this table must be stored in the folder **TNC:\table**.

The **toolturn.trn** turning tool table provides the following parameters:

Parameter	Meaning
T	Row number in the turning tool table The tool number allows identifying each tool unambiguously (e.g., for calling a tool). **Further information:** "Tool call by TOOL CALL", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Tool name?** The tool name identifies a tool, for example when calling it. **Further information:** "Tool call by TOOL CALL", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 Input: **Text width 32**
ZL 	**Tool length 1?** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**
XL 	**Tool length 2?** Length of the tool in the X direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**

Parameter	Meaning
YL 	**Tool length 3?** Length of the tool in the Y direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**
DZL 	**Oversize in tool length 1?** Delta value of tool length 1 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Oversize in tool length 2?** Delta value of tool length 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**
DYL 	**Tool length oversize 3?** Delta value of tool length 3 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Cutting edge radius?** The control takes into account the cutter radius for tool tip radius compensation. **Further information:** "Tooth radius compensation for turning tools (option 50)", Page 1117 In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 744 For the cutter geometry, the control also considers the parameters **TO**, **T-ANGLE**, and **P-ANGLE**. Input: **0...99999.9999**
DRS 	**Cutter radius oversize?** Delta value of cutter radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to the parameter **RS** Input: **-999.9999...+999.9999**

Parameter	Meaning
TO	**Tool orientation?** From the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction. This information is necessary, for example, for calculating the cutter radius compensation, milling cutter radius compensation, plunge angle, etc. **Further information:** "Tooth radius compensation for turning tools (option 50)", Page 1117 Refer to your machine manual. The control displays the tool orientations that are possible for each tool type. The machine manufacturer can change this assignment. In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 744 For the cutter geometry, the control also considers the parameters **RS**, **T-ANGLE**, and **P-ANGLE**. Input: **1...19**
SPB-INSERT	**Angular offset?** Angular offset for recessing tools Input: **-90.0...+90.0**
ORI	**Angle of spindle orientation?** Angle of tool spindle for aligning the turning tool Input: **-360.000...+360.000**
T-ANGLE	**Tool angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 744 For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **P-ANGLE**. Input: **0...179.999**
P-ANGLE	**Point angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 744 For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **T-ANGLE**. Input: **0...179.999**

Parameter	Meaning
CUTLENGTH  	**Cutting length of recessing tool** Length of the cutting edge of a turning or recessing tool The control monitors the length of the cutting edge in the turning cycles. If the cutting depth programmed in the turning cycle is greater than the length of the cutting edge defined in the tool table, then the control will display a warning and will automatically reduce the cutting depth. **Further information:** "Fundamentals of turning cycles", Page 760 Input: **0...99999.9999**
CUTWIDTH  	**Width of recessing tool** The control uses the width of a recessing tool for calculations within cycles. **Further information:** "Cycles for milling and turning", Page 744 Input: **0...99999.9999**
DCW 	**Oversize f. recessing tool width** Delta value of recessing tool width as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1589 Is added to parameter **CUTWIDTH** Input: **-99999.9999...+99999.9999**
TYPE 	**Type of turning tool** Depending on the selected turning tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the turning tools", Page 283 **Further information:** "Tool management ", Page 297 Selection by means of a selection window Input: **ROUGH**, **FINISH**, **THREAD**, **RECESS**, **BUTTON**, and **RECTURN**
WPL-DX-DIAM	**Compensation value for the workpiece diameter** Compensation value for the workpiece diameter with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 1018 Input: **-99999.9999...+99999.9999**
WPL-DZL	**Compensation value for the workpiece length** Compensation value for the workpiece length with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 1018 Input: **-99999.9999...+99999.9999**

Notes

- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- Geometry values from the tool table **tool.t**, such as length **L** or radius **R**, are not effective with turning tools.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
  - 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 least remaining tool life.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 2026
- The columns **WPL-DX-DIAM** and **WPL-DZL** are deactivated in the default configuration.

  In the machine parameter **columnKeys** (no. 105501), the machine manufacturer activates the columns **WPL-DX-DIAM** and **WPL-DZL**. The names of the columns may be different, however.

35.5.4 Grinding tool table toolgrind.grd (option 156)

Application

The grinding tool table **toolgrind.grd** contains the data specific to grinding tools.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 297
- Tool data required for grinding tools

  Further information: "Tool data for grinding tools (option 156)", Page 289
- Grinding operations on milling machines

  Further information: "Grinding operations (option 156)", Page 248
- Tool table for dressing tools

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 2019
- General tool data, regardless of the technology

  Further information: "Tool table tool.t", Page 1995

Requirements

- Jig grinding (software option 156)
- Grinding tool is defined in **TYPE** column of tool management
  Further information: "Tool types", Page 281

Description of function

NOTICE

Danger of collision!

In the tool management form, the control displays only the parameters relevant to the selected tool type. The tool tables contain locked parameters that are for internal consideration only. If you edit these additional parameters manually, tool data might no longer correctly match each other. There is a risk of collisions during subsequent movements!

- Edit the tools in the tool management form

NOTICE

Danger of collision!

The control differentiates between freely editable and locked parameters. The control writes to the locked parameters and uses these parameters for internal consideration. You must not manipulate these parameters. If you manipulate the locked parameters, tool data might no longer correctly match each other. There is a risk of collisions during subsequent movements!

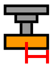
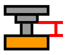
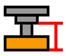
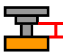
- Edit only freely editable tool management parameters
- Comply with the information about locked parameters in the tool data overview table

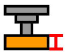
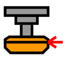
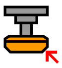
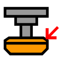
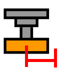
Further information: "Tool data for grinding tools (option 156)", Page 289

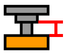
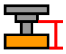
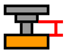
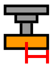
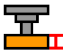
The file name of the grinding tool table is **toolgrind.grd** and this table must be stored in the folder **TNC:\table**.

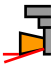
The **toolgrind.grd** grinding tool table provides the following parameters:

Parameter	Meaning
T	**Tool number** Row number in the grinding tool table The tool number allows identifying each tool unambiguously (e.g., for calling a tool). **Further information:** "Tool call", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 The row number must match the tool number in the **tool.t** tool table Input: **0...32767**

Parameter	Meaning
NAME	**Name of grinding wheel** The tool name identifies a tool, for example when calling it. **Further information:** "Tool call", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 Input: **Text width 32**
TYPE 	**Type of grinding wheel** Depending on the selected grinding tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the grinding tools", Page 283 **Further information:** "Tool management ", Page 297 Selection by means of a selection window Input: **GRIND_PIN**, **GRIND_CONE**, **GRIND_CUP**, **GRIND_CYLINDER**, **GRIND_ANGULAR** and **GRIND_FACE**
R-OVR 	**Radius of grinding wheel** Outermost radius of grinding tool After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.000000...999.999999**
L-OVR 	**Overhang of grinding wheel** Length up to the outermost radius of the grinding tool, with respect to the tool carrier reference point After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.000000...999.999999**
LO 	**Overall length** Absolute length of the grinding tool, with respect to the tool carrier reference point After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.000000...999.999999**
LI 	**Length to the inner edge** Length up to the inner edge, with respect to the tool carrier reference point After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.000000...999.999999**

Parameter	Meaning
B 	**Width** Width of the grinding tool After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.000000...999.999999**
G 	**Depth** Depth of grinding wheel After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.000000...999.999999**
ALPHA	**Angle for the slant** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.00000...90.00000**
GAMMA	**Angle for the corner** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **45.00000...180.00000**
RV 	**Radius at the edge for L-OVR** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.00000...999.99999**
RV1 	**Radius at the edge for LO** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.00000...999.99999**
RV2 	**Radius at the edge for LI** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962 Input: **0.00000...999.99999**
dR-OVR 	**Compensation of the radius** Delta value of the radius for tool compensation Is added to the parameter **R-OVR** Input: **-999.999999...+999.999999**

Parameter	Meaning
dL-OVR 	**Compensation of the overhang** Delta value of the overhang for tool compensation Is added to the parameter **L-OVR** Input: **-999.999999...+999.999999**
dLO 	**Compensation of the total length** Delta value of the total length for tool compensation Is added to the parameter **LO** Input: **-999.999999...+999.999999**
dLI 	**Compensation of the length to the inner edge** Delta value of the length up to the inner edge for tool compensation Is added to the parameter **LI** Input: **-999.999999...+999.999999**
R_SHAFT 	**Radius of the tool shank** Input: **0.00000...999.99999**
R_MIN 	**Min. permissible radius** If, after dressing, the actual radius is below the minimum permissible radius defined here, the control will display an error message. Input: **0.00000...999.99999**
B_MIN 	**Min. permissible width** If, after dressing, the actual width is below the minimum permissible width defined here, the control will display an error message. Input: **0.00000...999.99999**
V_MAX 	**Maximum permissible cutting speed** Cutting speed limit This value cannot be exceeded by programming a higher value or by using the potentiometer. Input: **0.000...999.999**
V	**Current cutting speed** Currently no function Input: **0.000...999.999**
W	**Tilt angle** Currently no function Input: **-90.00000...90.0000**
W_TYPE	**Tilted toward inner or outer edge** Currently no function Input: **-1**, **0**, **+1**
KIND	**Type of machining (internal/external grinding)** Currently no function Input: **0**, **1**
HW	**Wheel has a relief cut** Currently no function Input: **0**, **1**

Parameter	Meaning
HWA 	**Angle for relief cut on the outer edge** Input: **0.00000...45.00000**
HWI 	**Angle for relief cut on the inner edge** Input: **0.00000...45.00000**
INIT_D_OK	**Initial dressing performed** Initial dressing is the first dressing operation performed on the grinding wheel. Currently no function Input: **0**, **1**
INIT_D_PNR	**Dresser location for initial dressing** Dressing location used for initial dressing Input: **0...9999**
INIT_D_DNR	**Dresser number for initial dressing** Number of the dresser used for initial dressing Input: **0...32767**
MESS_OK	**Measure the grinding wheel** The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Input: **0**, **1**
STATE	**Setup status** Currently no function Input: **%0000000000000000...%1111111111111111**
A_NR_D	**Dresser number (diameter dressing)** The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Tool number of the dresser being used Corresponds to the **T_DRESS** parameter in the tool management Input: **0...32767**
A_NR_A	**Dresser number (outer edge dressing)** Currently no function Input: **0...32767**
A_NR_I	**Dresser number (inner edge dressing)** Currently no function Input: **0...32767**
DRESS_N_D 	**Dressing counter for diameter (specification)** Currently no function Input: **0...999**
DRESS_N_A 	**Dressing counter for outer edge (specification)** Currently no function Input: **0...999**

Parameter	Meaning
DRESS_N_I	**Dressing counter for inner edge (specification)** Currently no function Input: **0...999**
DRESS_N_D_ACT	**Current dressing counter of the diameter** Currently no function Input: **0...999**
DRESS_N_A_ACT	**Current dressing counter of the outer edge** Currently no function Input: **0...999**
DRESS_N_I_ACT	**Current dressing counter of the inner edge** Currently no function Input: **0...999**
AD	**Retraction amount at the diameter** The control uses this parameter when using a cycle for dressing. **Further information:** "General information on the dressing cycles", Page 916 Input: **0.00000...999.99999**
AA	**Retraction amount at the outer edge** The control uses this parameter when using a cycle for dressing. **Further information:** "General information on the dressing cycles", Page 916 Input: **0.00000...999.99999**
AI	**Retraction amount at the inner edge** The control uses this parameter when using a cycle for dressing. **Further information:** "General information on the dressing cycles", Page 916 Input: **0.00000...999.99999**
FORM	**Wheel shape** Currently no function Input: **0.00...99.99**
A_PL	**Chamfer length at outside** Currently no function Input: **0.00000...999.99999**
A_PW	**Chamfer angle at outside** Currently no function Input: **0.00000...89.99999**
A_R1	**Corner radius at outside** Currently no function Input: **0.00000...999.99999**
A_L	**Length of outside** Currently no function Input: **0.00000...999.99999**

Parameter	Meaning
A_HL	**Length of relief cut, wheel depth at outside** Currently no function Input: **0.00000...999.99999**
A_HW	**Angle of relief cut at outside** Currently no function Input: **0.00000...45.00000**
A_S	**Side depth at outside** Currently no function Input: **0.00000...999.99999**
A_R2	**Angle of departure at outside** Currently no function Input: **0.00000...999.99999**
A_G	**Reserve at outside** Currently no function Input: **0.00000...999.99999**
I_PL	**Chamfer length at inside** Currently no function Input: **0.00000...999.99999**
I_PW	**Chamfer angle at inside** Currently no function Input: **0.00000...89.99999**
I_R1	**Corner radius at inside** Currently no function Input: **0.00000...999.99999**
I_L	**Length of inside** Currently no function Input: **0.00000...999.99999**
I_HL	**Length of relief cut, wheel depth at inside** Currently no function Input: **0.00000...999.99999**
I_HW	**Angle of relief cut at inside** Currently no function Input: **0.00000...45.00000**
I_S	**Side depth at inside** Currently no function Input: **0.00000...999.99999**
I_R2	**Angle of departure at inside** Currently no function Input: **0.00000...999.99999**
I_G	**Reserve at inside** Currently no function Input: **0.00000...999.99999**

Parameter	Meaning
COR_ANG	**Inclination angle of dressing tool** Currently no function Input: **0.00000**...**360.00000**
COR_TYPE	**Selection of compensation method** You can choose between the following compensation methods: ■ **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** Compensation method removing material from the grinding tool **Further information:** "Stock removal on the grinding tool", Page 253 ■ **Dressing tool with wear, COR_TYPE_DRESSTOOL** Compensation method removing material from the dresser **Further information:** "Stock removal on the grinding tool", Page 253 Selection by means of a selection window Input: **0**, **1**

Notes

- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with grinding tools.
- When dressing a grinding tool, the tool must not be assigned a tool carrier kinematic model.
- Measure the grinding tool after dressing so that the control enters the correct delta values.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
  - 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 least remaining tool life.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 2026

35.5.5 Dressing tool table tooldress.drs (option 156)

Application

The dressing tool table **tooldress.drs** contains the data specific to dressing tools.

Related topics

- Editing tool data in tool management
  Further information: "Tool management ", Page 297
- Tool data required for dressing tools
  Further information: "Tool data for dressing tools (option 156)", Page 293
- Initial dressing
  Further information: "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 962
- Grinding operations on milling machines
  Further information: "Grinding operations (option 156)", Page 248
- Tool table for grinding tools
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 2010
- General tool data, regardless of the technology
  Further information: "Tool table tool.t", Page 1995

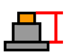
Requirements

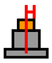
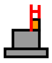
- Jig grinding (software option 156)
- Dressing tool is defined in **TYP** column of tool management
  Further information: "Tool types", Page 281

Description of function

The file name of the dressing tool table is **tooldress.drs** and this table must be stored in the folder **TNC:\table**.

The **tooldress.drs** dressing tool table provides the following parameters:

Parameter	Meaning
T	Row number in the dressing tool table The tool number allows identifying each tool unambiguously (e.g., for calling a tool). **Further information:** "Tool call by TOOL CALL", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Name of dressing tool** The tool name identifies a tool, for example when calling it. **Further information:** "Tool call by TOOL CALL", Page 304 You can define an index after the period. **Further information:** "Indexed tool", Page 276 Input: **Text width 32**
ZL 	**Tool length 1** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**

Parameter	Meaning
XL 	**Tool length 2** Length of the tool in the X direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**
YL 	**Tool length 3** Length of the tool in the Y direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 271 Input: **-99999.9999...+99999.9999**
DZL 	**Tool length oversize 1** Delta value of tool length 1 for tool compensation Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Tool length oversize 2** Delta value of tool length 2 for tool compensation Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**
DYL 	**Tool length oversize 3** Delta value of tool length 3 for tool compensation Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Tool tip radius** Input: **0.0000...99999.9999**
DRS 	**Cutter radius oversize** Delta value of the cutter radius for tool compensation Is added to the parameter **RS** Input: **-999.9999...+999.9999**
TO 	**Tool orientation** The control uses the tool orientation to determine the position of the tool's cutting edge. Input: **1...9**
CUTWIDTH	**Width of tool (plate, roll)** Tool width of the tool types **dressing plate** and **dressing roll** Input: **0.0000...99999.9999**
TYPE 	**Type of dressing tool** Depending on the selected dressing tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the dressing tools", Page 283 **Further information:** "Tool management ", Page 297 Selection by means of a selection window Input: **DRESS_FIX_RADIUS**, **HORNED**, **DRESS_ROT_RADIUS**, **DRESS_FIX_FLAT** and **DRESS_ROT_FLAT**

Parameter	Meaning
N-DRESS	**Speed of the tool (dressing spindle)** Shaft speed of a dressing spindle or dressing roll Input: **0.0000...99999.9999**

Notes

- The dressing tool will not be mounted to the spindle. You need to mount the dressing tool manually to a pocket defined by the machine manufacturer. Additionally, you must define the tool in the pocket table.
- When dressing a grinding tool, the tool must not be assigned a tool carrier kinematic model.

  Further information: "Pocket table tool_p.tch", Page 2026
- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with dressing tools.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
  - 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 least remaining tool life.
- If you want to archive tool tables, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 2026

35.5.6 Touch probe table tchprobe.tp

Application

The touch probe table **tchprobe.tp** defines the touch probe and data for the probing process, such as the probing feed rate. If you use several touch probes, you can save separate data for each touch probe.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 297
- Touch probe functions

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557
- Programmable touch probe cycles

  Further information: "Programmable Touch Probe Cycles", Page 1589

Description of function

NOTICE

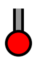
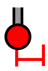
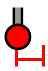
Danger of collision!

The control cannot use Dynamic Collision Monitoring (DCM) to protect L-shaped styluses against collisions. When using a touch probe with an L-shaped stylus there is a risk of collision!

- Carefully run in the NC program or program section in the **Program Run Single Block** operating mode
- Watch out for possible collisions!

The file name of the touch probe table is **tchprobe.tp** and this table must be stored in the folder **TNC:\table**.

The touch probe table **tchprobe.tp** provides the following parameters:

Parameter	Meaning
NO	**Sequential number of touch probe** You use this number to assign the touch probe to the data in the tool management column **TP_NO**. Input: **1...99**
TYPE 	**Selection of the touch probe?** The TS 642 touch probe makes the following values available: ■ **TS642-3**: The touch probe is activated by a conical switch. This mode is not supported. ■ **TS642-6**: The touch probe is activated by an infrared signal. Select this mode. Input: **TS120**, **TS220**, **TS249**, **TS260**, **TS440**, **TS444**, **TS460**, **TS630**, **TS632**, **TS640**, **TS642-3**, **TS642-6**, **TS649**, **TS740**, **TS 760**, **KT130**, **OEM**
CAL_OF1 	**TS center misalignmt. ref. axis? [mm]** According to the selection of the **STYLUS** column, this parameter has the following function: ■ **SIMPLE**: Offset of the touch probe axis to the spindle axis in the main axis ■ **L-TYPE**: Length of extension on an L-shaped stylus Input: **-99999.9999...+99999.9999**
CAL_OF2 	**TS center misalignmt. aux. axis? [mm]** Offset of the touch probe axis to the spindle axis in the secondary axis Input: **-99999.9999...+99999.9999**
CAL_ANG 	**Spindle angle for calibration?** According to the selection of the **STYLUS** column, this parameter has the following function: ■ **SIMPLE**: Prior to calibrating or probing, the control orients the touch probe with this spindle angle (if possible). ■ **L-TYPE**: The control orients the extension using the spindle angle. Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible). Input: **0.0000...359.9999**

Parameter	Meaning
F	**Probing feed rate? [mm/min]** In the machine parameter **maxTouchFeed** (no. 122602), the machine manufacturer defines the maximum probing feed rate. If **F** is greater than the maximum probing feed rate, then the maximum probing feed rate will be used. Input: **0...9999**
FMAX	**Rapid traverse in probing cycle? [mm/min]** Feed rate at which the control pre-positions the touch probe and positions it between the measuring points Input: **+10...+99999**
DIST	**Maximum measuring range? [mm]** If the stylus is not deflected in a probing process within the defined value, the control will display an error message. Input: **0.00100...99999.99999**
SET_UP	**Set-up clearance? [mm]** Distance of touch probe from the defined touch point when pre-positioning The smaller this value is, the more exactly you must define the touch point position. Safety clearances defined in the touch probe cycle are added to this value. Input: **0.00100...99999.99999**
F_PREPOS	**Pre-position at rapid? ENT/NOENT** Speed for pre-positioning: ■ **FMAX_PROBE**: Pre-position at the speed from **FMAX** ■ **FMAX_MACHINE**: Pre-position at machine rapid traverse Input: **FMAX_PROBE**, **FMAX_MACHINE**
TRACK	**Probe oriented? Yes=ENT/No=NOENT** Orienting the infrared touch probe in each probing process: ■ **ON**: The control orients the touch probe in the defined probing direction. In this way, the stylus is always deflected in the same direction, improving measuring accuracy. ■ **OFF**: The control will not orient the touch probe. If you change the **TRACK** parameter, you must recalibrate the touch probe. Input: **ON**, **OFF**
SERIAL	**Serial number?** The control automatically edits this parameter of touch probes with an EnDat interface. Input: **Text width 15**
REACTION	**Reaction? EMERGSTOP=ENT/NCSTOP=NOENT** As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. Reaction to resetting the ready signal: ■ **NCSTOP**: Interrupt NC program ■ **EMERGSTOP**: Emergency stop, quick braking of the axes Input: **NCSTOP**, **EMERGSTOP**

Parameter	Meaning
STYLUS	**Shape of the stylus** ■ **SIMPLE**: Straight stylus ■ **L-TYPE**: L-shaped stylus

Editing the touch probe table

To edit the touch probe table:

- ▶ Select the **Tables** operating mode
- ▶ Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- ▶ Select the **tchprobe.tp** file in the **Open File** workspace
- Open ▶ Select **Open**
- > The control opens the **Touch probes** application.
- Edit ▶ Activate **Edit**
- ▶ Select the desired value
- ▶ Edit the value

Notes

- You can also edit the touch probe table values in the tool management.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- In the machine parameter **overrideForMeasure** (no. 122604), the machine manufacturer defines whether you will be allowed to change the feed rate with the feed-rate potentiometer during probing.

35.5.7 Creating a tool table in inches

To create a tool table in inches:

- Select the **Manual** operating mode
- Select **T**
- Select the tool **T0**
- Press the **NC Start** key
- > The control removes the current tool and does not insert a new tool.
- Restart the control
- Do not acknowledge **Power interrupted**
- Select the **Files** operating mode
- Open the **TNC:\table** folder
- Rename the original file (e.g., **tool.t** as **tool_mm.t**)
- Select the **Tables** operating mode
- Select **Add**
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select a folder with a corresponding file extension (e.g., **t**)
- Select the desired prototype
- Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter a name (e.g., **tool**)
- Select **Create**
- > The control opens the **Tool table** tab in **Tables** operating mode.
- Restart the control
- Acknowledge **Power interrupted** with the **CE** key
- Select the **Tool table** tab in **Tables** operating mode
- > The control uses the newly created table as a tool table.

35.6 Pocket table tool_p.tch

Application

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine. The control needs the pocket table in order to change the tool.

Related topics

- Tool call

  Further information: "Tool call", Page 304
- Tool table

  Further information: "Tool table tool.t", Page 1995

Requirement

- The tool is defined in the tool management.
  Further information: "Tool management ", Page 297

Description of function

The file name of the pocket table is **tool_p.tch** and this table must be stored in the folder **TNC:\table**.

The **tool_p.tch** pocket table provides the following parameters:

Parameter	Meaning
P	**Pocket number?** Pocket number of the tool in the tool magazine Input: **0.0...99.9999**
T	**Tool number?** Row number of the tool from the tool table **Further information:** "Tool table tool.t", Page 1995 Input: **1...99999**
TNAME	**Tool name?** Name of the tool from the tool table When you define the tool number, the control will automatically load the tool name. **Further information:** "Tool table tool.t", Page 1995 Input: **Text width 32**
RSV	**Reserve pocket?** When a tool is in the spindle, the control reserves the pocket of this tool in the box magazine. To reserve the pocket for the tool: ■ No value entered: Pocket is not reserved ■ **R**: Pocket is reserved Input: No value, **R**
ST	**Special tool?** Define the tool as a special tool (e.g., with oversize tools): ■ No value entered: No special tool ■ **S**: Special tool Input: No value, **S**
F	**Fixed pocket?** Always return the tool to the same pocket in the tool magazine (e.g., with special tools) To define a fixed pocket for the tool: ■ No value entered: No fixed pocket ■ **F**: Fixed pocket Input: No value, **F**

Parameter	Meaning
L	**Locked pocket?** To lock a pocket for tools (e.g., the pockets next to special tools): ■ No value entered: Do not lock ■ **L**: Lock Input: No value, **L**
DOC	**Pocket comment?** The control automatically loads the tool comment from the tool table. **Further information:** "Tool table tool.t", Page 1995 Input: **Text width 32**
PLC	**PLC status?** Information about this tool pocket, which is transferred to the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **%00000000...%11111111**
P1 ... P5	**Value?** The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **-99999.9999...+99999.9999**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **0...99**
LOCKED_ABOVE	**Lock pocket above?** Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_BELOW	**Lock pocket below?** Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_LEFT	**Lock pocket at left?** Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_RIGHT	**Lock pocket at right?** Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LAST_USE	**LAST_USE** The control automatically loads the date and time of the last tool call from the tool table. **Further information:** "Tool table tool.t", Page 1995 Refer to your machine manual. Entry: **Text width 20**

Parameter	Meaning
S1	**S1** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**
S2	**S2** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**

35.7 Tool usage file

Application

The control saves information about the tools of an NC program in a tool usage file (e.g., all the required tools and the tool usage times). The control needs this file for the tool usage test.

Related topics

- Using the tool usage test
  Further information: "Tool usage test", Page 312
- Working with pallet tables
  Further information: "Pallet Machining and Job Lists", Page 1937
- Tool data from the tool table
  Further information: "Tool table tool.t", Page 1995

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer
  The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.
  Further information: "Creating the tool usage file", Page 313
- The **Generate tool-usage file** function setting is set to either **once** or **always**
  Further information: "Channel settings", Page 2104

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file Input: **0...99999**
TOKEN	In the **TOKEN** column, the control uses one word to show which information is contained in the respective row: ■ **TOOL**: Data per tool call; listed in chronological order ■ **TTOTAL**: All data of a tool; listed in alphabetical order ■ **STOTAL**: Called NC programs; listed in chronological order ■ **TIMETOTAL**: Total tool usage time of an NC program ■ **TOOLFILE**: Path of the tool table This enables the control during the tool usage test to detect whether you have performed the simulation with the tool table **tool.t** Input: **Text width 17**
TNR	Tool number If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767**
IDX	Tool index Input: **0...9**
NAME	Tool name Input: **Text width 32**
TIME	Tool usage time in seconds Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Input: **0...9999999**
WTIME	Total tool usage time in seconds Total time between the tool changes, during which the tool is cutting a workpiece Input: **0...9999999**
RAD	Sum of the tool radius **R** and the delta radius **DR** from the tool table Input: **-999999.9999...999999.9999**
BLOCK	NC block number of the tool call Input: **0...999999999**
PATH	Path of the NC program, the pallet table, or the tool table Input: **Text width 300**
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767.9**

Parameter	Meaning
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value **100**. Input: **0...32767**
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value **-1**. Input: **-1...32767**
NAMEPRG	Type of tool definition during a tool call: ■ **0**: The tool number is programmed ■ **1**: The tool name is programmed Input: **0**, **1**
LINENR	Row number of the pallet table in which the NC program is defined Input: **-1...99999**

Note

The control creates dependency files (***.dep**); for example, the tool-usage file in order to perform a tool usage test.

In the machine parameter **dependentFiles** (no. 122101) the machine manufacturer defines whether the control displays dependency files.

35.8 T usage order (option 93)

Application

In the **T usage order** table, the control displays the tool call sequence in an NC program. Before starting the program, you can see, for example, when a manual tool change will take place.

Requirements

- Extended Tool Management (software option 93)
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 313

  Further information: "Tool usage file", Page 2029

Description of function

When you select an NC program in **Program Run** operating mode, the control will automatically create the **T usage order** table. The control displays the table in the **T usage order** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot modify the table.

The **T usage order** table provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows
T	Number of the tool used, including an index as needed **Further information:** "Indexed tool", Page 276 May differ from the programmed tool (e.g., when a replacement tool is used)
NAME	Name of the tool used, including an index as needed **Further information:** "Indexed tool", Page 276 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ **OK**: Tool is in order ■ **Locked**: Tool is locked ■ **Not found**: Tool is not defined in the pocket table **Further information:** "Pocket table tool_p.tch", Page 2026 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool management ", Page 297
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 276
USAGE	Total tool usage time from the **WTIME** column of the **tool usage file** (in seconds) Total time between the tool changes, during which the tool is cutting a workpiece **Further information:** "Tool usage file", Page 2029
TOOL TIME	Estimated time of tool change
M3/M4 TIME	Tool usage time from the **TIME** column of the **tool usage file** (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) **Further information:** "Tool usage file", Page 2029
MIN OVRD	Minimum value of the feed-rate potentiometer during program run (in percent)
MAX OVRD	Maximum value of the feed-rate potentiometer during program run (in percent)
NC PGM	Path of the NC program in which the tool is programmed
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. **Further information:** "Pocket table tool_p.tch", Page 2026

35.9 Tooling list (option 93)

Application

In the **Tooling list** table, the control displays information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine.

Requirements

- Extended Tool Management (software option 93)
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 313

  Further information: "Tool usage file", Page 2029

Description of function

When you select an NC program in **Program Run** operating mode, the control will automatically create the **Tooling list** table. The control displays the table in the **Tooling list** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot modify the table.

The **Tooling list** table provides the following parameters:

Parameter	Meaning
T	Number of the tool used, including an index as needed **Further information:** "Indexed tool", Page 276 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ **OK**: Tool is in order ■ **Locked**: Tool is locked ■ **Not found**: Tool is not defined in the pocket table **Further information:** "Pocket table tool_p.tch", Page 2026 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool carrier management", Page 301
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 276
M3/M4 TIME	Tool usage time from the **TIME** column of the **tool usage file** (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) **Further information:** "Tool usage file", Page 2029
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. **Further information:** "Pocket table tool_p.tch", Page 2026

35.10 Freely definable tables

Application

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.

Related topics

- Variable functions **FN 26** to **FN 28**

  Further information: "NC functions for freely definable tables", Page 1394

Description of function

When you create a freely definable table, the control will provide various table templates for selection.

The machine manufacturers can create their own table templates and store them in the control.

35.10.1 Creating freely definable tables

To create a freely definable table:

- Select the **Tables** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **tab** folder
- Select the desired prototype

Select a path

- Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter the desired name

Create

- Select **Create**
- > The control opens the table.
- Modify the table as needed

  Further information: "Table workspace", Page 1982

Note

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: "Table access with SQL statements", Page 1417

35.11 Preset table

Application

The **preset.pr** preset table allows you to manage presets, such as the position and misalignment of a workpiece in the machine. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 210

Related topics

- Setting and activating presets

  Further information: "Preset management", Page 1025

Description of function

The preset table is stored in the **TNC:\table** directory by default and is named **preset.pr**. In the **Tables** operating mode, the preset table is open by default.

> Refer to your machine manual.
>
> The machine manufacturer can define a different path for the preset table.
>
> In the optional machine parameter **basisTrans** (no. 123903), the machine manufacturer defines a specific preset table for each range of traverse.

Icons and buttons of the preset table

The preset table contains the following icons:

Icon	Meaning
	Active row
	Write-protected row

When you define a preset, the control opens a window with the following input options:

Icon or button	Function
	**actual position capture** The control opens or closes the position display of the status overview. When you select an axis, the control applies the selected value at **Enter new**. **Further information:** "actual position capture in the preset table", Page 2040
Enter new	The control interprets the entered value as desired display value for the actual position. The control calculates the required table value from this. The entered value is active in the basic coordinate system **B-CS**. **Further information:** "Basic coordinate system B-CS", Page 1014 When you activate the edited preset, the control displays the entered value as actual position in the position display.
Correct	The control offsets the entered value against the actual table value. You can enter either a positive or a negative value. The entered value is active incrementally in the basic coordinate system **B-CS**.
Edit	The control accepts the entered value unchanged as table value. The entered value refers to the coordinate origin of the basic coordinate system **B-CS**.

Parameters of the preset table

The preset table contains the following parameters:

Parameter	Meaning
NO	Number of preset table row Input: **0...99999999**
DOC	Comment Entry: **Text width 16**
X	X coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 1014 Input: **-99999.99999...+99999.99999**
Y	Y coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 1014 Input: **-99999.99999...+99999.99999**
Z	Z coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 1014 Input: **-99999.99999...+99999.99999**
SPA	Spatial angle of preset in the A axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a 3D basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 1014 Input: **-99999.9999999...+99999.9999999**
SPB	Spatial angle of preset in the B axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a 3D basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 1014 Input: **-99999.9999999...+99999.9999999**
SPC	Spatial angle of preset in the C axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 1014 Input: **-99999.9999999...+99999.9999999**
X_OFFS	Position of the X axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.99999...+99999.99999**
Y_OFFS	Position of the Y axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.99999...+99999.99999**
Z_OFFS	Position of the Z axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.99999...+99999.99999**

Parameter	Meaning
A_OFFS	Axis angle of the A axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.9999999...+99999.9999999**
B_OFFS	Axis angle of the B axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.9999999...+99999.9999999**
C_OFFS	Axis angle of the C axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.9999999...+99999.9999999**
U_OFFS	Position of the U axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.99999...+99999.99999**
V_OFFS	Position of the V axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.99999...+99999.99999**
W_OFFS	Position of the W axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 1012 Input: **-99999.99999...+99999.99999**
ACTNO	Active workpiece preset The control automatically enters **1** in the active row. Input: **0**, **1**
LOCKED	Write-protection of the table row Entry: **Text width 16**

Refer to your machine manual.

In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.

Basic transformation and offset

The control interprets the basic transformations **SPA**, **SPB** and **SPC** as basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. During program execution, the control moves the linear axes in accordance with the basic rotation without any change in the workpiece position.

Further information: "Basic rotation and 3D basic rotation", Page 1027

The control interprets all offsets for each respective axis as a shift in the machine coordinate system **M-CS**. The effect that offsets have is contingent on the kinematics.

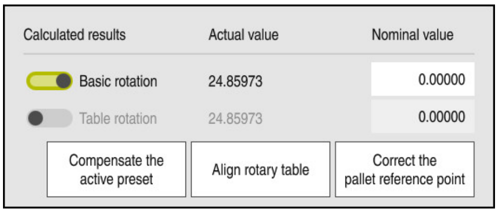
Further information: "Machine coordinate system M-CS", Page 1012

> HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

Application example

Use the **Rotation (ROT)** probing function to determine the misalignment of a workpiece. You can transfer the result to the preset table either as a basic transformation or as an offset.

Further information: "Determining and compensating the rotation of a workpiece", Page 1569



Results of the **Rotation (ROT)** probing function**Rotation (ROT)**

If you activate the **Basic rotation** toggle switch, the control interprets the misalignment as a basic transformation. When using the **Compensate the active preset** button, the control saves the result in the columns **SPA**, **SPB** and **SPC** of the preset table. The **Align rotary table** button has no function in this case.

If you activate the **Table rotation** toggle switch, the control interprets the misalignment as an offset. When using the **Compensate the active preset** button, the control saves the result in the columns **A_OFFS**, **B_OFFS** and **C_OFFS** of the preset table. To move the rotary axes to the position of the offset, use the **Align rotary table** button.

Write-protection for table rows

The **Lock record** button allows protecting any rows of the preset table against overwriting. The control enters the value **L** into the **LOCKED** column.

Further information: "Protecting table rows without a password", Page 2041

Alternatively, the row can be protected with a password. The control enters the value **###** into the **LOCKED** column.

Further information: "Protecting table rows with a password", Page 2041

The control displays an icon ahead of write-protected rows.

> If the control displays the value **OEM** in the **LOCKED** column, this column has been locked by the machine manufacturer.

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

35.11.1 actual position capture in the preset table

To load the actual position of an axis into the preset table:

- Activate the **Edit** toggle switch
- Double-tap or double-click the table row to be changed (e.g., in the **X** column)
- > The control opens a window with input options.

- Select **actual position capture**
- > The control opens the position display of the status overview.
- Select the desired value
- > The control loads the value into the window and activates the **Enter new** button.

- Select **OK**
- > The control calculates the table value that is needed and enters the value in the table.
- If required, close the position display of the status overview

35.11.2 Activating write protection

Protecting table rows without a password

To protect a table row without a password:

- Activate the **Edit** toggle switch
- Select the desired row
- Activate the **Lock record** toggle switch
- > The control enters the value **L** in the **LOCKED** column.
- > The control activates write-protection and displays an icon ahead of the row.

Protecting table rows with a password

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

To protect a table row with a password:

- Activate the **Edit** toggle switch
- Double-tap or double-click the **LOCKED** column of the desired row
- Enter the password
- Confirm your input
- > The control enters the value **###** in the **LOCKED** column.
- > The control activates write-protection and displays an icon ahead of the row.

35.11.3 Removing write protection

Unlocking table rows that are protected without a password

To unlock a table row that is protected without a password:

- Activate the **Edit** toggle switch
- Deactivate the **Lock record** toggle switch
- > The control removes the value **L** from the **LOCKED** column.
- > The control deactivates the write protection and removes the icon ahead of the row.

Unlocking table rows that are protected with a password

NOTICE
Caution: Data may be lost!
Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.
▶ Protecting table rows without a password is recommended ▶ Note down your passwords

To unlock a table row that is protected with a password:

- Activate the **Edit** toggle switch
- Double-tap or double-click the **LOCKED** column of the desired row
- Delete **###**
- Enter the password
- Confirm your input
- > The control deactivates write-protection and removes the icon ahead of the row.

35.11.4 Creating a preset table in inches

If you define inches as the unit of measure in the machine parameter **unitOfMeasure** (no. 101101), the unit of measure of the preset table will not be adjusted automatically.

To create a preset table in inches:

- Select the **Files** operating mode
- Open the **TNC:\table** folder
- Rename the **preset.pr** file (e.g., as **preset_mm.pr**)
- Select the **Tables** operating mode
- Select **Add**
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **pr** folder
- Select the desired prototype
- Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter the name **preset.pr**
- Select **Create**
- > The control opens the **Presets** tab in **Tables** operating mode.
- Restart the control
- Select the **Presets** tab in **Tables** operating mode
- > The control uses the newly created table as a preset table.

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.
▶ Before activating a preset, check whether all columns contain values.

- To optimize the file size and the processing speed, keep the preset table as short as possible.
- New rows can be inserted only at the end of the preset table.
- If you edit the value of the **DOC** column, you must reactivate the preset. Only then will the control accept the new value.

  Further information: "Activating presets", Page 1026
- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

  Further information: "Pallet preset table", Page 1951

Notes about machine parameters

- The machine manufacturer uses the optional machine parameter **initial** (no. 105603) to define a default value for each column of a new row.
- If the unit of measure of the preset table does not match the unit of measure defined in the machine parameter **unitOfMeasure** (no. 101101), the control displays a message in the dialog bar in **Tables** operating mode.
- In the optional machine parameter **presetToAlignAxis** (no. 300203), the machine manufacturer defines for each axis how the control interprets offsets for the following NC functions:
  - **FUNCTION PARAXCOMP**

    Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1284
  - **FUNCTION POLARKIN** (option 8)

    Further information: "Machining with polar kinematics with FUNCTION POLARKIN", Page 1295
  - **FUNCTION TCPM** or **M128** (option 9)

    Further information: "Compensating for the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1104
  - **FACING HEAD POS** (option 50)

    Further information: "Using a facing slide with FACING HEAD POS (option 50)", Page 1291

35.12 Point table

Application

In a point table, you save randomly distributed points on a workpiece. The control calls a cycle at each point. You can hide individual points and define a clearance height.

Related topics

- Calling point tables, effect with different cycles

  Further information: "Point tables", Page 400

Description of function

Parameters in point tables

The point table provides the following parameters:

Parameter	Meaning
NR	Row number in the point table Input: **0...99999**
X	X coordinate of a point Input: **-99999.9999...+99999.9999**
Y	Y coordinate of a point Input: **-99999.9999...+99999.9999**
Z	Z coordinate of a point Input: **-99999.9999...+99999.9999**
FADE	**Hide? (yes=ENT/no=NO ENT)** **Y**=**Yes:** The point is hidden during machining. Points that have been hidden will remain hidden until they are manually shown again. **N**=**No:** The point is shown for machining. All points of a point table are shown for machining by default. Input: **Y**, **N**
CLEARANCE	**Clearance height?** Safe position in the tool axis to which the control retracts the tool after machining a point. If you do not define a value in the **CLEARANCE** column, the control will use the value of the cycle parameter **Q204 2ND SET-UP CLEARANCE**. If you have defined values in both the **CLEARANCE** column and the **Q204** parameter, the control will use the higher of the two values. Input: **-99999.9999...+99999.9999**

35.12.1 Creating a point table

To create a point table:

- Select the **Tables** operating mode
- Select **Add**
- The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- The control opens the **Create new table** window.
- Select the **pnt** folder
- Select the desired prototype

Select a path

- Select **Select a path**
- The control opens the **Save as** window.
- Select the **table** folder
- Enter the desired name

Create

- Select **Create**
- The control opens the point table.

> 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:** "Table access with SQL statements", Page 1417

35.12.2 Hiding individual points during machining

In the **FADE** column of the point table, you can specify if the defined point will be hidden during the machining process.

To hide points:

- Select the desired point in the table
- Select the **FADE** column

Edit

- Activate **Edit**
- Enter **Y**
- The control hides the point at the cycle call.

If you enter **Y** in the **FADE** column, you can use the **Skip block** toggle switch to skip this point in the **Program Run** operating mode.

Further information: "Icons and buttons", Page 1956

35.13 Datum table

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. The datums can be called from within an NC program, for example in order to execute machining processes on several workpieces at the same position. The active row of the datum table serves as the workpiece datum in the NC program.

Related topics

- Contents and creation of a datum table
  Further information: "Datum table", Page 2045
- Editing a datum table during a program run
  Further information: "Compensation during program run", Page 1974
- Preset table
  Further information: "Preset table", Page 2035

Description of function

Parameters in datum tables

A datum table provides the following parameters:

Parameter	Meaning
D	Row number in the datum table Input: **0...99999999**
X	X coordinate of the datum Input: **-99999.99999...+99999.99999**
Y	Y coordinate of the datum Input: **-99999.99999...+99999.99999**
Z	Z coordinate of the datum Input: **-99999.99999...+99999.99999**
A	A coordinate of the datum Input: **-360.0000000...+360.0000000**
B	B coordinate of the datum Input: **-360.0000000...+360.0000000**
C	C coordinate of the datum Input: **-360.0000000...+360.0000000**
U	U coordinate of the datum Input: **-99999.99999...+99999.99999**
V	V coordinate of the datum Input: **-99999.99999...+99999.99999**
W	W coordinate of the datum Input: **-99999.99999...+99999.99999**
DOC	**Comment on shift?** Input: **Text width 15**

35.13.1 Creating a datum table

To create a datum table:

- Select the **Tables** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **d** folder
- Select the desired prototype
- Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter the desired name
- Select **Create**
- > The control opens the datum table.

> 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:** "Table access with SQL statements", Page 1417

35.13.2 Editing a datum table

You can edit the active datum table during program run.

Further information: "Compensation during program run", Page 1974

To edit a datum table:

- Activate **Edit**
- Select the value
- Edit the value
- Save the edited value, for example by selecting a different row

NOTICE

Danger of collision!

The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.

- Make sure to confirm any changes made to the table immediately (e.g., by pressing the **ENT** key)
- Activate the datum or compensation value in the NC program again
- Carefully test the NC program after changing the table values

35.14 Tables for cutting data calculation

Application

The following tables allow you to calculate the cutting data of a tool in the cutting data calculator:

- Table for workpiece materials **WMAT.tab**
  Further information: "Table for workpiece materials WMAT.tab", Page 2048
- Table for tool materials **TMAT.tab**
  Further information: "Table for tool materials TMAT.tab", Page 2048
- Cutting data table ***.cut**
  Further information: "Cutting data table *.cut", Page 2049
- Diameter-dependent cutting data table ***.cutd**
  Further information: "Diameter-dependent cutting data table *.cutd", Page 2050

Related topics

- Cutting data calculator
  Further information: "Cutting data calculator", Page 1529
- Tool management
  Further information: "Tool management ", Page 297

Description of function

Table for workpiece materials WMAT.tab

In the table for workpiece materials **WMAT.tab**, you define the workpiece material. You must save this table in the **TNC:\table** folder.

The table for workpiece materials **WMAT.tab** provides the following parameters:

Parameter	Meaning
WMAT	Workpiece material (e.g., aluminum) Input: **Text width 32**
MAT_CLASS	Material class Categorize the materials into material classes with the same cutting conditions, e.g., in accordance with DIN EN 10027-2. Input: **Text width 32**

Table for tool materials TMAT.tab

In the table for tool materials **TMAT.tab**, you define the tool material. You must save this table in the **TNC:\table** folder.

The table for tool materials **TMAT.tab** provides the following parameters:

Parameter	Meaning
TMAT	Tool material (e.g., solid carbide) Input: **Text width 32**
ALIAS1	Additional designation Input: **Text width 32**
ALIAS2	Additional designation Input: **Text width 32**

Cutting data table *.cut

In the cutting data table ***.cut**, you assign the matching cutting data to the workpiece materials and the tool materials. You must save the table in the **TNC:\system\Cutting-Data** folder.

The cutting data table ***.cut** provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows Input: **0...999999999**
MAT_CLASS	Workpiece material from the **WMAT.tab** table **Further information:** "Table for workpiece materials WMAT.tab", Page 2048 Selection by means of a selection window Input: **0...9999999**
MODE	Machining mode (e.g., roughing or finishing) Input: **Text width 32**
TMAT	Tool material from the table **TMAT.tab** **Further information:** "Table for tool materials TMAT.tab", Page 2048 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 309 Input: **0...1000**
FTYPE	Type of feed: ■ **FU**: Feed per revolution **FU** in mm/rev ■ **FZ**: Feed per tooth **FZ** in mm/tooth **Further information:** "Feed rate F", Page 310 Input: **FU**, **FZ**
F	Feed rate value Input: **0.0000...9.9999**

Diameter-dependent cutting data table *.cutd

In the diameter-dependent cutting data table ***.cutd**, you assign the matching cutting data to the workpiece materials and the tool materials. You must save the table in the **TNC:\system\Cutting-Data** folder.

The diameter-dependent cutting data table ***.cutd** provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows Input: **0...999999999**
MAT_CLASS	Workpiece material from the **WMAT.tab** table **Further information:** "Table for workpiece materials WMAT.tab", Page 2048 Selection by means of a selection window Input: **0...9999999**
MODE	Machining mode (e.g., roughing or finishing) Input: **Text width 32**
TMAT	Tool material from the table **TMAT.tab** **Further information:** "Table for tool materials TMAT.tab", Page 2048 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 309 Input: **0...1000**
FTYPE	Type of feed: ■ **FU**: Feed per revolution **FU** in mm/rev ■ **FZ**: Feed per tooth **FZ** in mm/tooth **Further information:** "Feed rate F", Page 310 Input: **FU**, **FZ**
F_D_0...**F_D_9999**	Feed rate value for the respective diameter You don't need to define all columns. If a tool diameter is between two defined columns, the control linearly interpolates the feed rate. Input: **0.0000...9.9999**

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.

35.15 Pallet table

Application

Pallet tables allow you to define the sequence in which the control will machine the pallets and the NC programs to be used.

Without a pallet changer, you can use pallet tables to successively run NC programs with different presets with just one press of **NC Start**. This type of usage is also called job list.

Tool-oriented machining is possible with pallet tables and with job lists. The control will reduce the number of tool changes, thereby reducing the machining time.

Related topics

- Editing and executing a pallet table in the **Job list** workspace
  Further information: "Job list workspace", Page 1938
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 1948

Requirement

- Pallet Management (software option 22)

Description of function

Pallet tables can be opened in **Tables**, **Editor**, and **Program Run** operating modes. In **Editor** and **Program Run** operating modes, the control opens the pallet table in the **Job list** workspace instead of as a table.

The machine manufacturer defines a prototype for the pallet table. When you create a new pallet table, the control will copy this prototype. This means that the pallet table on your control might not contain all possible parameters.

The prototype can include the following parameters:

Parameter	Meaning
NR	Row number in the pallet table The entry is required for the **Line number** input field of the **BLOCK SCAN** function. **Further information:** "Block scan for mid-program startup", Page 1965 Input: **0...99999999**
TYPE	**Pallet type?** Contents of the table row: ■ **PAL**: Pallet ■ **FIX**: Fixture ■ **PGM**: NC program Selection using a selection menu Input: **PAL**, **FIX**, **PGM**
NAME	**Pallet / NC program / Fixture?** File name of the pallet, fixture or NC program The machine manufacturer specifies the names of pallets and fixtures as needed. You can define the names of your NC programs yourself. Selection by means of a selection window Input: **Text width 32**

Parameter	Meaning
DATUM	**Datum table?** The datum table to be used in the NC program. Selection by means of a selection window Input: **Text width 32**
PRESET	**Preset?** Row number in the preset table for the workpiece preset to be activated. Selection by means of a selection window Input: **0...999**
LOCATION	**Location?** The entry **MA** indicates that there is a pallet or fixture in the working space of the machine and can be machined. Press the **ENT** key to enter **MA**. Press the **NO ENT** key to remove the entry and thus suppress machining. If the column exists, the entry is mandatory. Corresponds to the **Machinable** toggle switch in the **Form** workspace. Selection using a selection menu Input: No value, **MA**
LOCK	**Locked?** Using an * you can exclude the row of the pallet table from execution. Press the **ENT** key to identify the row with the entry *. Press the **NO ENT** key to cancel the lock. You can lock the execution for individual NC programs, fixtures or entire pallets. Unlocked rows (e.g., PGM) in a locked pallet are also not executed. Selection using a selection menu Input: No value, *
W STATUS	**Machining status?** Relevant to tool-oriented machining The machining status defines the machining progress. Enter BLANK for an unmachined (raw) workpiece. The control changes this entry automatically during machining. The control differentiates between the following entries ■ BLANK / no entry: Workpiece blank, requires machining ■ INCOMPLETE: Partly machined, requires further machining ■ ENDED: Machined completely, no further machining required ■ EMPTY: Empty space, no machining required ■ SKIP: Skip machining **Further information:** "Tool-oriented machining", Page 1948 Input: No value, **BLANK**, **INCOMPLETE**, **ENDED**, **EMPTY**, **SKIP**
PALPRES	**Pallet preset** Row number in the pallet preset table for the pallet preset to be activated Only required if a pallet preset table has been created on the control. Selection by means of a selection window Input: **-1...+999**
DOC	Comment Input: **Text width 15**

Parameter	Meaning
METHOD	**Machining method?** Machining method The control differentiates between the following entries ■ WPO: Workpiece oriented (standard) ■ TO: Tool oriented (first workpiece) ■ CTO: Tool oriented (further workpieces) **Further information:** "Tool-oriented machining", Page 1948 Selection using a selection menu Input: **WPO**, **TO**, **CTO**
CTID	**ID no. geometry context?** Relevant to tool-oriented machining The control automatically generates the ID number for mid-program startup with block scan. If you delete or change the entry, mid-program startup is no longer possible. **Further information:** "Tool-oriented machining", Page 1948 Input: **Text width 8**
SP-X	**Clearance height?** Clearance height in the X axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
SP-Y	**Clearance height?** Clearance height in the Y axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
SP-Z	**Clearance height?** Clearance height in the Z axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
SP-A	**Clearance height?** Clearance height in the A axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
SP-B	**Clearance height?** Clearance height in the B axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
SP-C	**Clearance height?** Clearance height in the C axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
SP-U	**Clearance height?** Clearance height in the U axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**

Parameter	Meaning
SP-V	**Clearance height?** Clearance height in the V axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
SP-W	**Clearance height?** Clearance height in the W axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1948 Input: **-999999.99999...+999999.99999**
COUNT	**Number of operations** For rows of the **PAL** type: Current actual value for the pallet counter nominal value defined in the **TARGET** column. For rows of the **PGM** type: Value indicating by how much the pallet counter actual value will be incremented after the execution of the NC program. **Further information:** "Pallet counter", Page 1938 Input: **0...99999**
TARGET	**Total number of operations** Nominal value for the pallet counter in rows of the **PAL** type The control repeats the NC programs of this pallet until the nominal value has been reached. **Further information:** "Pallet counter", Page 1938 Input: **0...99999**

35.15.1 Creating and opening a pallet table

To create a pallet table:

- Select the **Tables** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **p** folder
- Select the desired prototype
- Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter the desired name
- Select **Create**
- > The control opens the table in **Tables** operating mode.

- The file name of a pallet table must always begin with a letter.
- Use the **Select in Program Run** button in **Files** operating mode to open the pallet table in **Program Run** operating mode. In this operating mode, you can edit and execute pallet tables.

  Further information: "Job list workspace", Page 1938

35.16 Compensation tables

35.16.1 Overview

The control provides the following compensation tables:

Table	Further information
Compensation table ***.tco** Compensation in the tool coordinate system **T-CS**	Page 2055
Compensation table ***.wco** Compensation in the working plane coordinate system **WPL-CS**	Page 2057

35.16.2 Compensation table *.tco

Application

The compensation table ***.tco** allows you to define compensation values for the tool in the tool coordinate system **T-CS**.

You can use the compensation table ***.tco** for tools of all types of technologies.

Related topics

- Using compensation tables

  Further information: "Tool compensation with compensation tables", Page 1120
- Contents of the compensation table ***.wco**

  Further information: "Compensation table *.wco", Page 2057
- Editing compensation tables during program run

  Further information: "Compensation during program run", Page 1974
- Tool coordinate system **T-CS**

  Further information: "Tool coordinate system T-CS", Page 1022

Description of function

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 compensation table ***.tco** provides the following parameters:

Parameter	Meaning
NO	Row number in the table Input: **0...999999999**
DOC	Comment Input: **Text width 16**
DL	**Tool length oversize?** Delta value for parameter **L** of the tool table Input: **-999.9999...+999.9999**
DR	**Tool radius oversize?** Delta value for parameter **R** of the tool table Input: **-999.9999...+999.9999**
DR2	**Tool radius oversize 2?** Delta value for parameter **R2** of the tool table Input: **-999.9999...+999.9999**
DXL	**Oversize in tool length 2?** Delta value for parameter **DXL** of the turning tool table Input: **-999.9999...+999.9999**
DYL	**Tool length oversize 3?** Delta value for parameter **DYL** of the turning tool table Input: **-999.9999...+999.9999**
DZL	**Oversize in tool length 1?** Delta value for parameter **DZL** of the turning tool table Input: **-999.9999...+999.9999**
DL-OVR	**Compensation of the overhang** Delta value for parameter **L-OVR** of the grinding tool table Input: **-999.9999...+999.9999**
DR-OVR	**Compensation of the radius** Delta value for parameter **R-OVR** of the grinding tool table Input: **-999.9999...+999.9999**
DLO	**Compensation of the total length** Delta value for parameter **LO** of the grinding tool table Input: **-999.9999...+999.9999**
DLI	**Compensation of the length to the inner edge** Delta value for parameter **LI** of the grinding tool table Input: **-999.9999...+999.9999**

35.16.3 Compensation table *.wco

Application

The values from the compensation tables with the ***.wco** file name extension are applied as shifts in the working plane coordinate system (**WPL-CS**).

The ***.wco** compensation tables are used mainly for turning (option 50).

Related topics

- Using compensation tables
  Further information: "Tool compensation with compensation tables", Page 1120
- Contents of the compensation table ***.tco**
  Further information: "Compensation table *.tco", Page 2055
- Editing compensation tables during program run
  Further information: "Compensation during program run", Page 1974
- Working plane coordinate system **WPL-CS**
  Further information: "Working plane coordinate system WPL-CS", Page 1018

Description of function

The compensation table ***.wco** provides the following parameters:

Parameter	Meaning
NO	Row number in the table Input: **0...999999999**
DOC	Comment Input: **Text width 16**
X	Shift of the working plane coordinate system **WPL-CS** in **X** Input: **-999.9999...+999.9999**
Y	Shift of **WPL-CS** in **Y** Input: **-999.9999...+999.9999**
Z	Shift of **WPL-CS** in **Z** Input: **-999.9999...+999.9999**

35.16.4 Creating a compensation table

To create a compensation table:

- Select the **Tables** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the folder **tco** or **wco**
- Select the desired prototype
- Select a path: Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter the desired name
- Create: Select **Create**
- > The control opens the table.

35.17 *.3DTC compensation table

Application

In a ***.3DTC** compensation table, the control saves the radius deviation of ball-nose cutters from the nominal value at a defined inclination angle. For workpiece touch probes, the control saves the deflection behavior of the touch probe at a defined probing angle.

The control takes into account the saved data during the execution of NC programs and during probing.

Related topics

- 3D radius compensation depending on the tool's contact angle

  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1140
- 3D calibration of the touch probe

  Further information: "Calibrating the workpiece touch probe", Page 1572

Requirements

- Advanced Functions Set 2 (software option 9)
- 3D-ToolComp (software option 92)

Description of function

The ***.3DTC** compensation tables must be saved in the **TNC:\system\3D-ToolComp** folder. In the **DR2TABLE** tool management column, you can then assign the tables to a tool.

You create a separate table for each tool.

A compensation table provides the following parameters:

Parameter	Meaning
NR	Sequential row number in the compensation table The control evaluates a maximum of 100 rows in the compensation value table. Input: **0...9999999**
ANGLE	Inclination angle of tools or probing angle of workpiece touch probes Input: **-99999.999999...+99999.999999**
DR2	Radius deviation from the nominal value or deflection of the touch probe Input: **-99999.999999...+99999.999999**

35.18 Tables for AFC (option 45)

35.18.1 Basic AFC settings in AFC.tab

Application

In the **AFC.TAB** table, you can enter the feed rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

Related topics

- Programming AFC
  Further information: "Adaptive Feed Control (AFC, option 45)", Page 1196

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

The data in this table are default values that are copied into a file belonging to the respective NC program during a teach-in cut. The values act as the basis for feedback control.

Further information: "Description of function", Page 2064

> If you define a tool-specific feedback-control reference power using the **AFC-LOAD** column in the tool table, the control generates the associated file for the relevant NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Parameter

The **AFC.tab** table provides the following parameters:

Parameter	Meaning
NR	Row number in the table Input: **0...9999**
AFC	Name of the control setting Enter this name in the **AFC** tool management column. It specifies the assignment of the control parameters to the tool. Input: Text width 10
FMIN	Feed rate at which the control will perform an overload response Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. **Further information:** "Monitoring tool wear and tool load", Page 1203 Input: **0...999**
FMAX	Maximum feed rate within the material up to which the control can automatically increase the feed rate Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. **Further information:** "Monitoring tool wear and tool load", Page 1203 Input: **0...999**
FIDL	Feed rate at which the control will traverse the tool outside of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: **0...999**
FENT	Feed rate at which the control will move the tool into and out of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: **0...999**

Parameter	Meaning
OVLD	Desired reaction of the control to overload: ■ **M**: Execution of a macro defined by the machine manufacturer ■ **S**: Immediate NC stop ■ **F:** NC stop once the tool has been retracted ■ **E**: Just display an error message on the screen ■ **L**: Disable active tool ■ **-**: No overload response If the maximum spindle power is exceeded for more than one second and the feed rate falls below the defined minimum during that time, the control will conduct an overload response. In conjunction with the cut-related tool wear monitoring function, the control will only evaluate the options **M**, **E**, and **L**! Input: **M**, **S**, **F**, **E**, **L**, or **-**
POUT	Spindle power at which the control will detect that the tool exits the workpiece Enter the value in percent of the learned reference load Recommended input value: 8% In turning mode: Minimum load **Pmin** for tool monitoring (option 50) Input: **0...100**
SENS	Sensitivity (aggressiveness) of feedback control 50 is for slow feedback control, 200 for a very aggressive feedback control. An aggressive feedback control responds quickly and significantly changes the values, but it tends to overshoot. In turning mode: Activate the monitoring of the minimum load **Pmin** (option 50): ■ **1**: Evaluate **Pmin** ■ **0**: Do not evaluate **Pmin** Input: **0...999**
PLC	Value that the control will transfer to the PLC at the beginning of a machining step The machine manufacturer defines whether and which function will be performed by the control. Input: **0...999**

Creating an AFC.tab table

You need to create the table only if the table is missing in the **table** folder.

To create the **AFC.tab** table:

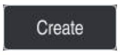
- ▶ Select the **Tables** operating mode

- ▶ Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.

- ▶ Select **Create new table**
- > The control opens the **Create new table** window.
- ▶ Select the **tab** folder

- ▶ Select the desired prototype

- ▶ Select **Select a path**
- > The control opens the **Save as** window.
- ▶ Select the **table** folder
- ▶ Enter the desired name



- ▶ Select **Create**
- > The control opens the table.

Notes

- If there is no AFC.TAB table in the **TNC:\table** directory, the control uses a permanently defined, internal control setting for the teach-in cut. If, alternatively, a tool-dependent reference power value exists, the control uses it immediately. HEIDENHAIN recommends using the AFC.TAB table in order to ensure safe and well-defined operation.
- 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: "Table access with SQL statements", Page 1417

35.18.2 AFC.DEP settings file for teach-in cuts

Application

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

Related topics

- AFC basic settings in the table **AFC.tab**

  Further information: "Basic AFC settings in AFC.tab", Page 2059
- Setting up and using AFC

  Further information: "Adaptive Feed Control (AFC, option 45)", Page 1196

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

Each row in the **<name>.H.AFC.DEP** file stands for a machining section, that you start with **FUNCTION AFC CUT BEGIN** and complete with **FUNCTION AFC CUT END**. You can edit all data of the **<name>.H.AFC.DEP** file for optimization purposes. If you have optimized the values from the AFC.TAB table, the control places a ***** in front of these control settings in the AFC column.

Further information: "Basic AFC settings in AFC.tab", Page 2059

In addition to the contents from the **AFC.tab** table, the **AFC.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed (not editable)
IDX	Index of the tool with which the machining step was performed (not editable)
N	Difference for tool call: ■ **0**: Tool was called by its tool number ■ **1**: Tool was called by its tool name
PREF	Reference load of the spindle. The control measures the value in percent with respect to the rated spindle power
ST	Status of the machining step: ■ **L**: In the next program run, a teach-in cut will be recorded for this machining step. The control will overwrite any existing values in this line ■ **C**: The teach-in cut was completed successfully. The next program run can be conducted with automatic feed control
AFC	Name of the control setting

Notes

- Note that the **<name>.H.AFC.DEP** file is locked against editing as long as the NC program **<name>.H** is running.

  The control does not remove the editing lock until one of the following functions has been executed:

  - **M2**
  - **M30**
  - **END PGM**
- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

35.18.3 Log file AFC2.DEP

Application

The control stores various pieces of information for each machining step of a teach-in cut in the **<name>.H.AFC2.DEP<name>.I.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. During feedback control, the control updates the data and performs various evaluations.

Related topics

- Setting up and using AFC
  Further information: "Adaptive Feed Control (AFC, option 45)", Page 1196

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

The **AFC2.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed
IDX	Index of the tool with which the machining step was performed
SNOM	Nominal spindle speed [rpm]
SDIFF	Maximum difference of the spindle speed in % of the nominal speed
CTIME	Machining time (tool in effect)
FAVG	Average feed rate (tool in effect)
FMIN	Smallest occurring feed factor. The control shows the value as a percentage of the programmed feed rate
PMAX	Maximum recorded spindle power during machining. The control shows the value as a percentage of the spindle's rated power.
PREF	Reference load of the spindle. The control shows the value as a percentage of the spindle's rated power.
OVLD	Overload reaction performed by the control: ■ **M**: A macro defined by the machine manufacturer has been run ■ **S:** Immediate NC stop was conducted ■ **F**: NC stop was conducted after the tool was retracted ■ **E**: An error message was displayed ■ **L**: The current tool was locked ■ **-**: There was no overload response
BLOCK	Block number at which the machining step begins

During feedback control, the control determines the current machining time as well as the resulting time saving in percent. The control enters the results of the evaluation between the key words **total** and **saved** in the last line of the log file. Where the time balance is positive, the percentage value is also positive.

Note

- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

35.18.4 Editing tables for AFC

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

To open a table for AFC:

- Select the **Program Run** operating mode

AFC settings

- Select **AFC settings**
- The control displays a selection menu. The control shows all existing tables for this NC program.
- Select a file (e.g., **AFC.TAB**)
- The control opens the file in the **Tables** operating mode.

35.19 Technology table for Cycle 287 Gear Skiving

Application

In Cycle **287 GEAR SKIVING**, you can use the cycle parameter **QS240 NUMBER OF CUTS** to call a table containing technology data. The table is a freely definable table, and as such is in ***.tab** format. The control provides you with a template. In the table, you define the following data for each individual cut:

- Feed rate
- Lateral infeed
- Lateral offset

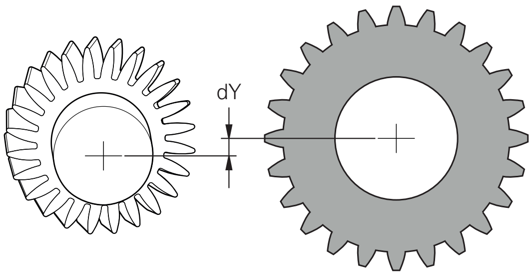
Requirements

- Software option 157: Gear Cutting

35.19.1 Parameters in the technology table

Parameter in the table

The technology data table contains the following parameters:

Parameter	Function
NR	Number of the cut that also corresponds to the number of the table row
FEED	Feed rate in mm/rev or 1/10 inch/rev for the cut This parameter replaces the following cycle parameters: ■ **Q588 FIRST FEED RATE** ■ **Q589 LAST FEED RATE** ■ **Q580 FEED-RATE ADAPTION** Input: **0...9999.999**
INFEED	Lateral infeed of the cut. This entry is incremental. This parameter replaces the following cycle parameters: ■ **Q586 FIRST INFEED** ■ **Q587 LAST INFEED** Input: **0...99.99999**
dY	Lateral offset of the cut (to improve the removal of chips). Input: **-9.99999...+9.99999** 

Notes

- The unit used in the NC program determines whether millimeter or inch units are used.
- In order to avoid contour distortions, HEIDENHAIN recommends that you do not program an offset **dY** in the last cut.
- HEIDENHAIN recommends that you program only minimum offset values **dY** in the individual cuts, because this might result in contour damage.
- The sum of the lateral infeeds (**INFEED**) must result in the tooth height.
  - If the tooth height is greater than the total infeed, the control will display a warning.
  - If the tooth height is less than the total infeed, the control will display an error message.

  Example:
  - **TOOTH HEIGHT** (**Q563**) = 2 mm
  - Number of cuts (**NR**) = 15
  - Lateral infeed (**INFEED**) = 0.2 mm
  - Total infeed = **NR** * **INFEED** = 3 mm

    In this case, the tooth height is less than the total infeed (2 mm < 3 mm).

    Reduce the number of cuts to 10.

35.19.2 Creating a technology table

To create a table containing technology data:

- ▶ Select the **Tables** operating mode
- ▶ Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- ▶ Select **Create new table**
- > The control opens the **Create new table** window.
- ▶ Select the **tab** folder
- ▶ Select the **Proto_Skiving.TAB** prototype
- ▶ Select **Select a path**
- > The control opens the **Save as** window.
- ▶ Select the **table** folder
- ▶ Enter the desired name
- ▶ Select **Create**
- > The control opens the technology table.

36

Electronic Handwheel

36.1 Fundamentals

Application

If you want to approach a position in the machine's working space while the guard door is open or if you execute a small infeed movement, you can use the electronic handwheel. The electronic handwheel allows you to traverse the axes and perform various functions provided by the control.

Related topics

- Incremental jog positioning

  Further information: "Incremental jog positioning of axes", Page 205
- Handwheel superimpositioning with GPS (option 44)

  Further information: "Function Handwheel superimp.", Page 1225
- Handwheel superimpositioning with **M118**

  Further information: "Activating handwheel superimpositioning with M118", Page 1334
- Virtual tool axis **VT**

  Further information: "Virtual tool axis VT", Page 1226
- Touch probe functions in **Manual** operating mode

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1557

Requirement

- Electronic handwheel (e.g., HR 550FS)

  The control supports the following electronic handwheels:
  - HR 410: Cable-bound handwheel without display
  - HR 420: Cable-bound handwheel with display
  - HR 510: Cable-bound handwheel without display
  - HR 520: Cable-bound handwheel with display
  - HR 550FS: Wireless handwheel with display, data transmission via radio

Description of function

You can use electronic handwheels in **Manual** or **Program Run** operating mode.

The HR 520 and HR 550FS portable handwheels feature a display that allows the control to show different types of information. You can use the handwheel soft keys for setup functions, such as the setting of presets or the activation of miscellaneous functions.

Once you have activated the handwheel with the handwheel activation key or the **Handwheel** toggle switch, you can operate the control only by using the handwheel. If you press the axis keys in this state, the control will display the message **Handwheel active: Handwheel-1, MB0**.

If more than one handwheel is connected to a control, you can activate or deactivate a handwheel only by pressing the handwheel activation key on the respective handwheel. You need to deactivate the active handwheel in order to be able to select another handwheel.

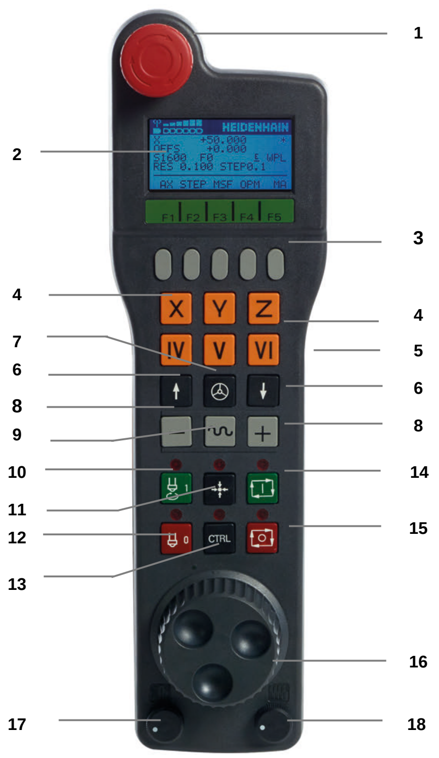
Functions in Program Run operating mode

You can perform the following functions in **Program Run** operating mode:

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel key)
- After the **NC Stop** key has been pressed: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After the **NC STOP** key has been pressed: Traverse manual axes (handwheel soft keys **MOP** and then **MAN**)
- Return to the contour after axes were manually traversed during an interruption of the program run (handwheel soft keys **MOP** and then **REPO**). The handwheel soft keys are used for operating.

  Further information: "Returning to the contour", Page 1972
- Switch on/off the “Tilt working plane” function (handwheel soft keys **MOP** and then **3D**)

Operating elements of an electronic handwheel



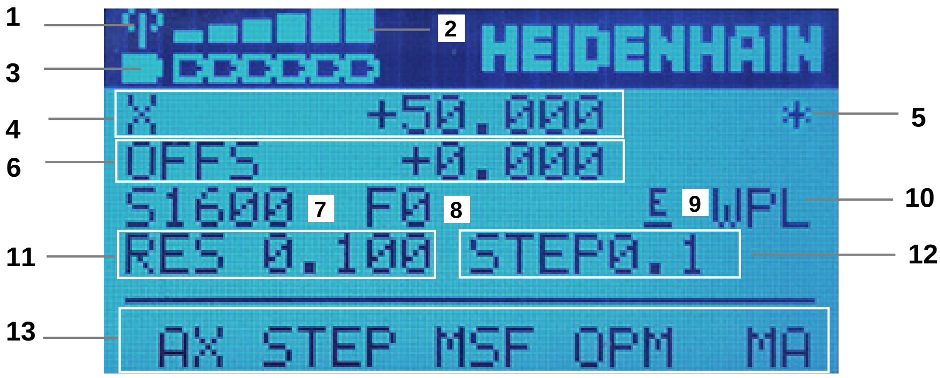
An electronic handwheel provides the following operating elements:

1 **EMERGENCY STOP** key
2 Handwheel display for status and for selecting functions
3 Handwheel soft keys
4 Axis keys; can be exchanged by the machine manufacturer depending on the axis configuration
5 Permissive button

  The permissive button is on the rear side of the handwheel.
6 Arrow keys for defining the handwheel resolution
7 Handwheel activation key

8 Axis-direction key
Key for the direction of the traverse motion
9 Rapid traverse override for the traverse motion
10 Spindle switch-on (machine-dependent function, key can be exchanged by the machine manufacturer)
11 **Generate NC block** key (machine-dependent function, key can be exchanged by the machine manufacturer)
12 Spindle switch-off (machine-dependent function, key can be exchanged by the machine manufacturer)
13 **CTRL** key for special functions (machine-dependent function, key can be exchanged by the machine manufacturer)
14 **NC Start** key (machine-dependent function, key can be exchanged by the machine manufacturer)
15 **NC Stop** key
Machine-dependent function; key can be exchanged by the machine manufacturer
16 Handwheel
17 Spindle speed potentiometer
18 Feed rate potentiometer
19 Cable connection; not required for the HR 550 FS wireless handwheel

Contents of an electronic handwheel display



The display of an electronic handwheel consists of the following areas:

1 Handwheel is in the docking station or radio mode is active
Only with HR 550 FS wireless handwheel
2 Field strength
Six bars = maximum field strength
Only with HR 550 FS wireless handwheel
3 Charging condition of battery
Six bars = maximum charging condition. A bar moves from the left to the right during recharging.
Only with HR 550 FS wireless handwheel
4 **X+50.000**: Position of the selected axis

5 *****: Control in operation; program run has been started or axis is in motion
6 Handwheel superimpositioning from **M118** or the Global Program Settings GPS (option 44)

   Further information: "Activating handwheel superimpositioning with M118", Page 1334

   Further information: "Function Handwheel superimp.", Page 1225
7 **S1000**: Current spindle speed
8 Feed rate at which the selected axis is moving

   The control displays the current contouring feed rate while the program is running.
9 **E**: Error message

   If an error message appears on the control, the handwheel display shows the message **ERROR** for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.
10 Active setting in the **3-D rotation** window:
   - **VT**: **Tool axis** function
   - **WP**: **Basic rotation** function
   - **WPL**: **3D ROT** function

   Further information: "3-D rotation window (option 8)", Page 1098
11 Handwheel resolution

   Distance that the selected axis moves per handwheel revolution

   Further information: "Handwheel resolution", Page 2074
12 Incremental jog active or inactive

   If the function is active, the control will display the active traverse step.
13 Soft-key row

   The soft key row provides the following functions:
   - **AX**: Select the machine axis

     Further information: "Creating a positioning block", Page 2076
   - **STEP**: Incremental jog positioning

     Further information: "Incremental jog positioning", Page 2076
   - **MSF:** Execute various functions of the **Manual** operating mode (e.g., entering the feed rate **F**)

     Further information: "Entering miscellaneous functions M", Page 2075
   - **OPM**: Select the operating mode
     - **MAN**: **Manual** operating mode
     - **MDI**: **MDI** application in **Manual** operating mode
     - **RUN**: **Program Run** operating mode
     - **SGL**: **Single Block** mode of **Program Run** operating mode
   - **MA**: Switching the magazine pockets

Handwheel resolution

The handwheel sensitivity specifies the distance an axis moves per handwheel revolution. The handwheel sensitivity results from the defined handwheel speed of the axis and the speed level used internally by the control. The speed level describes a percentage of the handwheel speed. The control calculates a specific handwheel sensitivity value for each speed level. The resulting handwheel sensitivity values are directly selectable with the handwheel arrow keys (only if incremental jog is not active).

The handwheel speed indicates the increment (e.g., 0.01 mm) traversed per handwheel detent position. You can change the handwheel speed with the handwheel's arrow keys.

If you have defined a handwheel speed of 1, the following handwheel resolutions are available:

Resulting handwheel sensitivity levels in mm/revolution and degrees/revolution:
0.0001/0.0002/0.0005/0.001/0.002/0.005/0.01/0.02/0.05/0.1/0.2/0.5/1

Resulting handwheel sensitivity levels in in/revolution:
0.000127/0.000254/0.000508/0.00127/0.00254/0.00508/0.0127/0.0254/0.0508/0.127/0.254/0.508

Examples for resulting handwheel sensitivity values:

Defined handwheel speed	Speed level	Resulting handwheel sensitivity
10	0.01%	0.001 mm/revolution
10	0.01%	0.001 degrees/revolution
10	0.0127%	0.00005 in/revolution

Effect of the feed-rate potentiometer when handwheel is active

NOTICE

Caution: Possible damage to the workpiece!

When toggling between the machine operating panel and the handwheel, the feed rate may be reduced. This can cause visible marks on the workpiece.

- Make sure to retract the tool before toggling between the handwheel and the machine operating panel.

The settings of the feed-rate potentiometer on the handwheel may differ from those on the machine operating panel. When you activate the handwheel, the control automatically activates the feed-rate potentiometer of the handwheel. When you deactivate the handwheel, the control automatically activates the feed-rate potentiometer of the machine operating panel.

In order to make sure that the feed rate does not increase while you switch over between the potentiometers, the feed rate is either frozen or reduced.

If the feed rate before switching over is higher than the feed rate after switching over, the control automatically reduces the feed rate to the smaller value.

If the feed rate before switching over is less than the feed rate after switching over, the control automatically freezes the feed rate. In this case, you must turn the feed-rate potentiometer back to the previous value because the activated feed-rate potentiometer will only then be effective.

36.1.1 Entering spindle speed S

To enter the spindle speed **S** by using an electronic handwheel:

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F2** (**S**)
- Select the desired speed by pressing the **F1** or **F2** key
- Press the **NC Start** key
- > The control activates the entered spindle speed.

> If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0.
>
> By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

36.1.2 Entering the feed rate F

To enter the feed rate **F** by using an electronic handwheel:

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F3** (**F**)
- Select the desired feed rate by pressing the **F1** or **F2** key
- Load the new feed rate F with the **F3** (**OK**) handwheel soft key

> If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0.
>
> By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

36.1.3 Entering miscellaneous functions M

To enter a miscellaneous function by using an electronic handwheel:

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F1** (**M**)
- Select the desired M function number by pressing the **F1** or **F2** key
- Press the **NC Start** key
- > The control activates the miscellaneous function

Further information: "Overview of miscellaneous functions", Page 1319

36.1.4 Creating a positioning block

Refer to your machine manual.

Your machine manufacturer can assign any function to the **Generate NC block** handwheel key.

To create a positioning block by using an electronic handwheel:

- Select the **Manual** operating mode
- Select the **MDI** application
- If necessary, select the NC block after which the positioning block should be inserted
- Activate the handwheel
- Press the **Generate NC block** key on the handwheel
- > The control inserts a straight line **L**, including all of the axis positions.

36.1.5 Incremental jog positioning

Incremental jog positioning allows you to move the selected axis by a preset value.

To incrementally position an axis by using an electronic handwheel:

- Press the handwheel soft key F2 (**STEP**)
- Press the handwheel soft key 3 (**ON**)
- > The control activates incremental jog positioning.
- Set the desired jog increment by using the **F1** or **F2** keys

The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches).

- Confirm the selected jog increment by pressing the handwheel soft key F4 (**OK**)
- Use the **+** or **-** handwheel key to move the active handwheel axis in the corresponding direction
- > The control moves the active axis by the entered increment every time the handwheel key is pressed.

If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0.

By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Notes

DANGER

Caution: hazard to the user!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- Devices should be connected or removed only by authorized service technicians
- Only switch on the machine via a connected handwheel or a secured connection

NOTICE

Caution: Danger to the tool and workpiece!

The wireless handwheel triggers an emergency stop reaction if the radio transmission is interrupted, the battery is fully empty, or if there is a defect. Emergency stop reactions during machining can cause damage to the tool or workpiece.

- Place the handwheel in the handwheel holder when it is not in use
- Keep the distance between the handwheel and the handwheel holder small (pay attention to the vibration alarm)
- Test the handwheel before machining

- The machine manufacturer can provide additional functions for the HR5xx handwheels.

  Refer to your machine manual.
- You can use the axis keys to activate the **X**, **Y**, and **Z** axes, as well as three other axes that can be defined by the machine manufacturer. Your machine manufacturer can also place the virtual axis **VT** on one of the free axis keys.

36.2 HR 550FS wireless handwheel

Application

With the HR 550FS wireless handwheel and its radio transmission characteristics, you can move farther away from the machine operating panel than with other handwheels. The HR 550FS wireless handwheel thus provides an important benefit, in particular for large machines.

Description of function

The HR 550FS wireless handwheel comes fitted with a rechargeable battery. The battery starts charging when you place the handwheel into the holder.

The HRA 551FS handwheel holder and the HR 550FS handwheel together form one function unit.

HR 550FS handwheel

HRA 551FS handwheel holder

The HR 550FS handwheel can be operated by battery for up to eight hours before it needs recharging. A completely discharged handwheel takes approx. three hours for a full charge. When you do not use the HR 550FS, always place it into the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.

When the handwheel is in its holder, it provides the same functionality as during radio mode. This allows you to use a completely discharged handwheel.

> Clean the contacts of the handwheel holder and handwheel regularly to ensure their proper functioning.
>
> If the control has triggered an emergency stop, you must reactivate the handwheel.
>
> **Further information:** "Reactivating the handwheel", Page 2082

If you happen to get close to the limit of the transmission range, the HR 550FS will set off a vibrating alarm. If this occurs, you must reduce the distance to the handwheel holder.

Note

⚠ DANGER

Caution: hazard to the user!

Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.

- Check the radio connection of the handwheel for possible overlapping with other wireless devices
- Switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test when it is restarted
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

36.3 Configuration of wireless handwheel window

Application

The **Configuration of wireless handwheel** window shows the connection data of the HR 550FS wireless handwheel and provides various functions for optimizing the radio connection, such as setting the radio channel.

Related topics

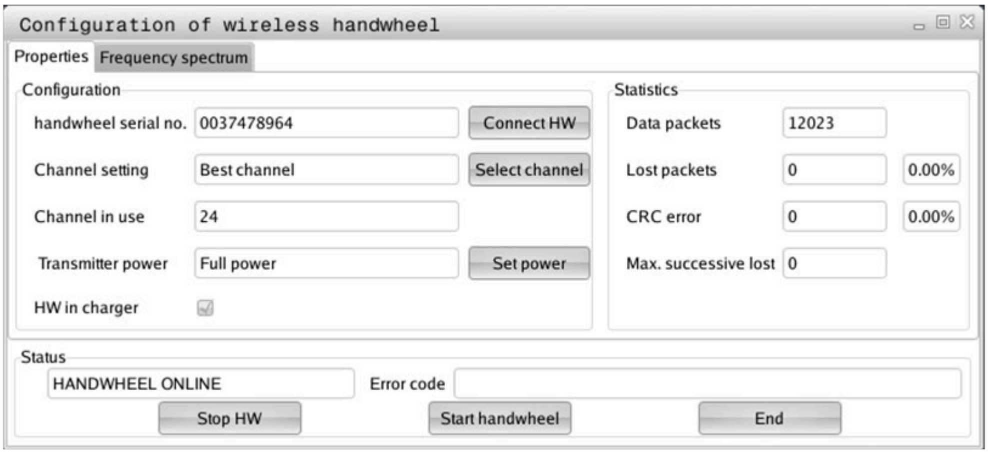
- Electronic handwheel

  Further information: "Electronic Handwheel", Page 2069
- HR 550FS wireless handwheel

  Further information: "HR 550FS wireless handwheel", Page 2078

Description of function

Use the **Set up wireless handwheel** menu item to open the **Configuration of wireless handwheel** window. The menu item is in the **Machine settings** group of the **Settings** application.



Areas of the Configuration of wireless handwheel window

Configuration area

In the **Configuration** area, the control displays different types of information about the connected wireless handwheel, such as the serial number.

Statistics area

In the **Statistics** area, the control displays information about the transmission quality.

If the received signal quality is impaired and no longer ensures a perfect, safe stop of the axes, the wireless handwheel will perform an emergency stop.

A high value under **Max. successive lost** is an indication of a limited quality of reception. If the control repeatedly displays values greater than 2 during normal operation of the wireless handwheel within the desired range of use, there is a high risk of undesired disconnection.

If this occurs, try to improve the transmission quality by selecting a different channel or by increasing the transmitter power.

Further information: "Setting the radio channel", Page 2082

Further information: "Selecting the transmission power", Page 2081

Status area

In the **Status** area, the control displays the current status of the handwheel, such as **HANDWHEEL ONLINE** and pending error messages concerning the connected handwheel.

36.3.1 Assigning a handwheel to a handwheel holder

In order to assign a handwheel to a handwheel holder, the handwheel holder must be connected to the control hardware.

To assign a handwheel to a handwheel holder:

- Place the handwheel into the handwheel holder
- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Machine settings** group
- Double-tap or double-click the **Set up wireless handwheel** menu item
- > The control opens the **Configuration of wireless handwheel** window.
- Select the **Connect HW** button
- > The control saves the serial number of the inserted wireless handwheel and displays it in the configuration window to the left of the **Connect HW** button.
- Select the **END** button
- > The control saves the configuration.

36.3.2 Selecting the transmission power

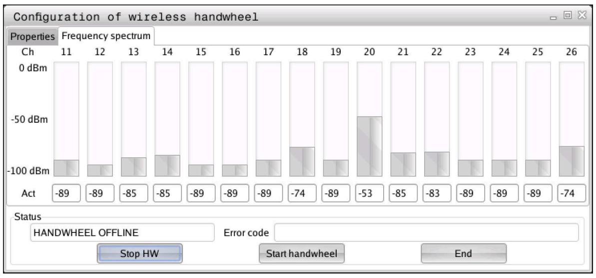
If you reduce the transmission power, the range of the wireless handwheel will decrease.

To set the transmission power of the handwheel:

- Open the **Configuration of wireless handwheel** window
- Select the **Set power** button
- > The control displays the three available power settings.
- Select the desired transmission power setting
- Select the **END** button
- > The control saves the configuration.

36.3.3 Setting the radio channel

If the wireless handwheel is started automatically, then the control tries to select the radio channel providing the best radio signal.



To set the radio channel manually:

- Open the **Configuration of wireless handwheel** window
- Select the **Frequency spectrum** tab
- Select the **Stop HW** button
- > The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.
- Note the number of the channel with the least amount of radio traffic

> The smallest bar indicates the channel with the least amount of radio traffic.

- Select the **Start handwheel** button
- > The control restores the connection to the wireless handwheel.
- Select the **Properties** tab
- Select the **Select channel** button
- > The controls shows all available channel numbers.
- Select the number of the channel with the least amount of radio traffic
- Select the **END** button
- > The control saves the configuration.

36.3.4 Reactivating the handwheel

To reactivate the handwheel:

- Open the **Configuration of wireless handwheel** window
- Use the **Start handwheel** button to reactivate the wireless handwheel
- Select the **END** button

37

Touch Probes

37.1 Setting up touch probes

Application

The **Device configuration** window allows you to create and manage all the workpiece and tool touch probes of the control.

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

Related topics

- Creating a workpiece touch probe with cable or infrared transmission by using the touch probe table

  Further information: "Touch probe table tchprobe.tp", Page 2022
- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)

  Further information: "Machine parameters", Page 2152

Description of function

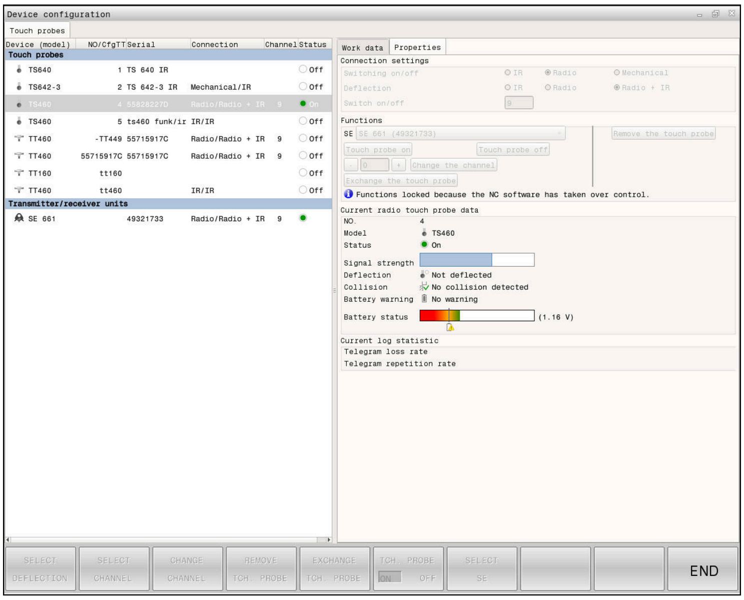
You open the **Device configuration** window in the **Machine settings** group of the **Settings** application. Double-tap or double-click the **Set up touch probes** menu item.

Further information: "Settings Application", Page 2099

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

In order for the control to recognize the touch probe with radio transmission, you will require an **SE 661** transceiver with EnDat interface.

You define the new values in the **Work data** area.



Areas of the Device configuration window

Touch probes area

In the **Touch probes** area, the control displays all of the defined workpiece and tool touch probes, as well as the transceiver units. All other areas provide detailed information about the selected entry.

Work data area

For a workpiece touch probe, the control displays the values from the touch probe table in the **Work data** area.

For a tool touch probe, the control displays the values from the machine parameter **CfgTT** (no. 122700).

You can select and edit the displayed values. Under **Touch probes**, the control displays information about the active value (e.g., selection options). You can change the values of the tool touch probes only after entering the code number 123.

Properties area

In the **Properties** area, the control displays the connection data and diagnostic functions.

For touch probes with radio connection, the control displays the following information in **Current radio touch probe data**:

Display	Meaning
NO.	Number in the touch probe table
Type	Type of touch probe
Status	Touch probe active or inactive
Signal strength	Display of the signal strength in the bar graphic The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision recognized
Battery status	Display of the battery quality If the charge is less than the displayed bar, then the control outputs a warning.

The **Switching on/off** connection setting is preset based on the type of touch probe. Under **Deflection**, you can select how the touch probe is to transmit the signal when probing.

Deflection	Meaning
IR	Infrared probe signal
Radio	Radio probe signal
Radio + IR	The control selects the probe signal

> If you activate the touch probe's radio connection by using the connection setting **Switch on/off**, then the signal will be retained even after a tool change. You need to use this connection setting to deactivate the radio connection.

Buttons

The control provides the following buttons:

Button	Function
CREATE TS ENTRY	Create a new workpiece touch probe You define the new values in the **Work data** area.
CREATE TT ENTRY	Create a new tool touch probe You define the new values in the **Work data** area.
SELECT DEFLECTION	Select the probe signal
SELECT CHANNEL	Select the radio channel Select the channel with the best radio transmission and pay attention to overlaps with other machines or wireless handwheels.
CHANGE CHANNEL	Change the radio channel
REMOVE TCH. PROBE	Delete the touch probe data The control deletes the entry from the **Device configuration** window and from the touch probe table or the machine parameters.
EXCHANGE TCH. PROBE	Save a new touch probe in the current row The control automatically overwrites the serial number of the replaced touch probe with the new number.
SELECT SE	Select the SE transceiver
SELECT IR POWER	Select the strength of the infrared signal You only need to change the signal strength if there is interference.
SELECT RADIO POWER	Select the strength of the radio signal You only need to change the signal strength if there is interference.

Note

In the machine parameter **CfgHardware** (no. 100102), the machine manufacturer defines whether the control will show or hide the touch probes in the **Device configuration** window. Refer to your machine manual.

38

Embedded Workspace and Extended Workspace

38.1 Embedded Workspace (option 133)

Application

You use Embedded Workspace to operate a Windows PC and display its screen contents on the control's user interface. You use Remote Desktop Manager (option 133) to connect the Windows PC.

Related topics

- Remote Desktop Manager (option 133)

  Further information: "Remote Desktop Manager window (option 133)", Page 2137
- Using Extended Workspace to operate a Windows PC through an additional connected monitor

  Further information: "Extended Workspace", Page 2090

Requirements

- Established RemoteFX connection to the Windows PC through Remote Desktop Manager (option 133)
- Connection defined in the machine parameter **CfgRemoteDesktop** (no. 133500)

  In the optional machine parameter **connections** (no. 133501), the machine manufacturer enters the name of the RemoteFX connection.

  Refer to your machine manual.

Description of function

Embedded Workspace is available on the control as an operating mode and as a workspace. If the machine manufacturer does not define a name, then the operating mode and workspace are both named **RDP**.

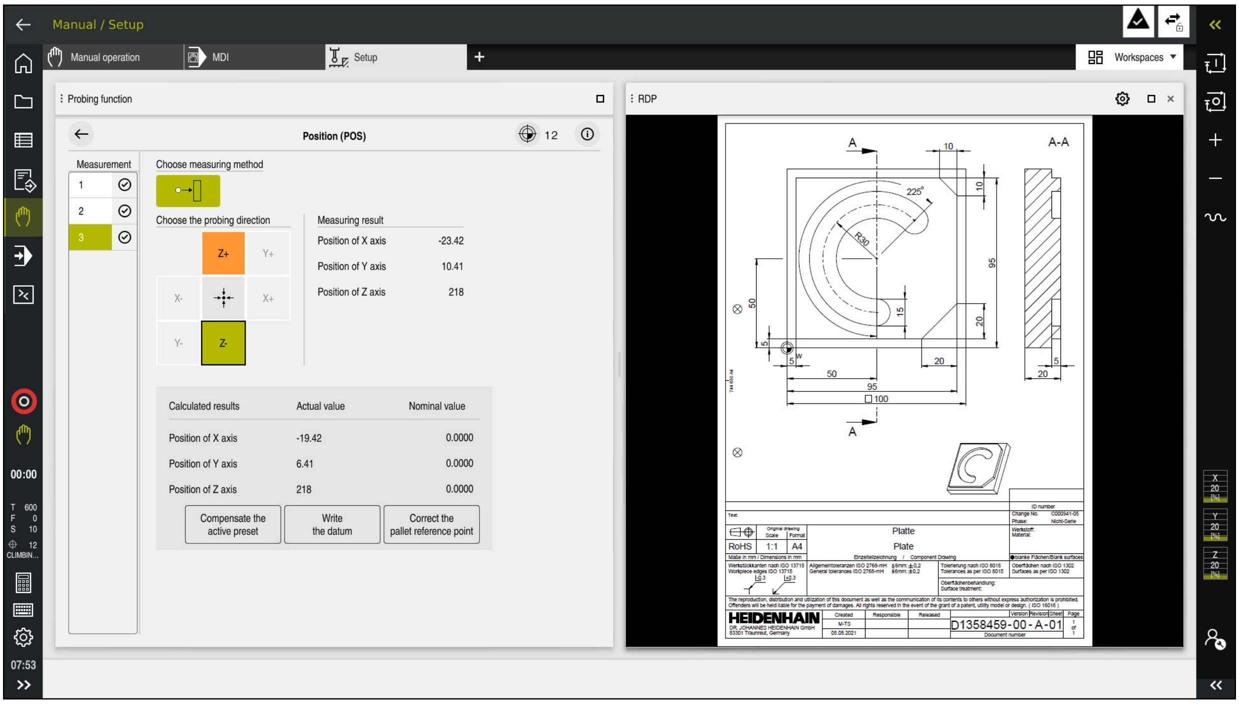
Entries cannot be made through the Windows PC as long as the RemoteFX connection is active. This avoids the problem of conflicting operation.

Further information: "Windows Terminal Service (RemoteFX)", Page 2138

If you open Embedded Workspace as an operating mode, the control displays a full-screen version of the Windows PC user interface in it.

If you open Embedded Workspace as a workspace, you can change the size and position of the workspace as you wish. The control rescales the user interface of the Windows PC after each modification.

Further information: "Workspaces", Page 112



Embedded Workspace as workspace with opened PDF file

RDP settings window

If Embedded Workspace is open as a workspace, you can open the **RDP settings** window.

The **RDP settings** window contains the following buttons:

Button	Meaning
Reconnect	If the control could not establish a connection to the Windows PC, for example due to a timeout, press this button to try again. The control can also display this button in the operating mode and the workspace.
Adjust resolution	With this button the control rescales the user interface of the Windows PC to the size of the workspace.

38.2 Extended Workspace

Application

With Extended Workspace you can use an additional attached monitor as a second screen of the control. That way you can use the additional monitor independently of the control's user interface and also to show the control's applications.

Related topics

- Using Embedded Workspace to operate a Windows PC within the control's user interface (option 133

  Further information: "Embedded Workspace (option 133)", Page 2088
- ITC hardware expansion

  Further information: "Hardware enhancements", Page 107

Requirement

- Additional attached monitor configured by the machine manufacturer as Extended Workspace

  Refer to your machine manual.

Description of function

Here are some functions you can perform with Extended Workspace:

- Opening files from the control (e.g., drawings)
- Opening windows from HEROS functions in addition to the control's user interface

  Further information: "HEROS menu", Page 2184
- Displaying and operating computers connected through Remote Desktop Managers (option 133)

  Further information: "Remote Desktop Manager window (option 133)", Page 2137

39

Integrated Functional Safety (FS)

Application

The safety concept of integrated functional safety (FS) for machines with HEIDENHAIN controls offers supplementary software safety functions in addition to the mechanical safety features of the machine. For example, the integrated safety concept automatically reduces the feed rate when you perform operations with open guard doors. The machine manufacturer can modify or expand the FS safety concept.

Requirements

- Integrated functional safety (FS, basic version; software option 160) or Integrated functional safety (FS, full version; software option 161)
- Software options 162 to 166 or software option 169, if necessary

  Whether you need these software options depends on the machine's number of drives.
- The machine manufacturer must adapt the FS safety concept to the machine.

Description of function

Every machine tool user is exposed to certain risks. While protective devices can prevent access to dangerous locations, the user must also be able to work on the machine without this protection (e.g. guard door opened).

Safety functions

To ensure that the requirements for operator protection are met, integrated functional safety (FS) provides standardized safety functions. The machine manufacturer uses the standardized safety functions for implementing functional safety (FS) for the machine in question.

You can track the active safety functions in the axis status of functional safety (FS).

Further information: "Axis status menu item", Page 2095

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of drives using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the drives
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the drives
SLS	Safely Limited Speed	Safely limited speed. Prevents the drives from exceeding the specified speed limits when the protective door is opened
SLP	Safely Limited Position	Safely limited position. Monitors safe axes to keep them within the limit values of a defined area
SBC	Safe Brake Control	Dual-channel control of the motor holding brakes

Safety-related operating modes of functional safety (FS)

Functional safety (FS) of a control offers various safety-related operating modes. The safety-related operating mode with the lowest number has the highest safety level.

Depending on how the machine manufacturer implements them, the following safety-related operating modes are available:

Refer to your machine manual.
The machine manufacturer must adapt the safety-related operating modes to each machine.

Icon	Safety-related operating mode	Short description
SOM 1	Operating mode **SOM_1**	Safe operating mode 1: Automatic mode, production mode
SOM 2	Operating mode **SOM_2**	Safe operating mode 2: Set-up mode
SOM 3	Operating mode **SOM_3**	Safe operating mode 3: Manual intervention; only for qualified operators
SOM 4	Operating mode **SOM_4** This function must be enabled and adapted by the machine manufacturer.	Safe operating mode 4: Advanced manual intervention, process monitoring, only for qualified users

Functional safety (FS) in the Positions workspace

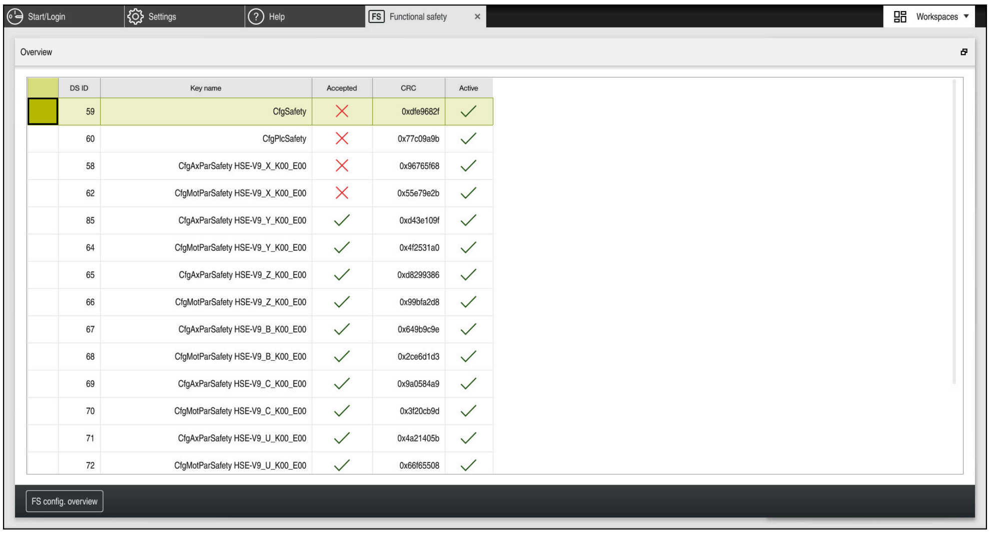
On a control with functional safety (FS), the control displays the monitored operating statuses of the speed **S** and feed rate **F** in the **Positions** workspace. If a safety function is triggered while in a monitored state, the control stops the feed movement and the spindle or reduces the speed (e.g., if a guard door is opened).

Further information: "Axis display and position display", Page 164

Functional safety application

Refer to your machine manual.
The machine manufacturer configures the safety functions in this application.

In the **Functional safety** application in the **Home** operating mode, the control provides information about the status of the individual safety functions. In this application you can see whether individual safety functions are active and have been accepted by the control.

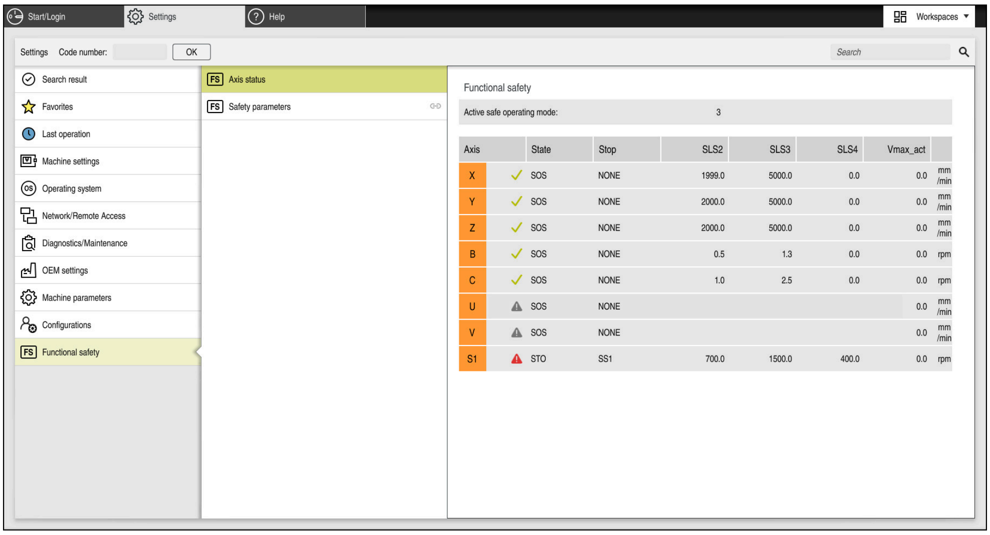


Functional safety application

Axis status menu item

In the **Axis status** menu item of the **Settings** application, the control provides the following information about the status of the individual axes:

Field	Meaning
Axis	Configured axes of the machine
State	Active safety function
Stop	Stop reaction **Further information:** "Functional safety (FS) in the Positions workspace", Page 2093
SLS2	Maximum speed or feed-rate values for **SLS** in the **SOM_2** operating mode
SLS3	Maximum speed or feed-rate values for **SLS** in the **SOM_3** operating mode
SLS4	Maximum speed or feed-rate values for **SLS** in the **SOM_4** operating mode This function must be enabled and adapted by the machine manufacturer.
Vmax_act	Currently valid speed or feed rate limit These are either values from the **SLS** settings or from the SPLC If values are greater than 999 999, the control displays **MAX**.



Axis status menu item in the **Settings** application

Test status of the axes

In order for the control to ensure safe operation of the axes, it checks all monitored axes when the machine is switched on.

The control checks whether the position of an axis matches the position directly after shutdown. If a deviation is detected, the control marks the respective axis in the position display with a red warning triangle.

If checking of individual axes fails when starting the machine, you can check the axes manually.

Further information: "Checking axis positions manually", Page 2097

The control indicates the test status of the individual axes with the following icons:

Icon	Meaning
✓	The axis has been tested or does not need to be tested.
⚠ (red)	The axis has not been tested, but must be tested to ensure safe operation. **Further information:** "Checking axis positions manually", Page 2097
⚠ (gray)	The axis is not monitored by functional safety (FS) or is not configured as a safe axis.

Feed-rate limiting with functional safety (FS)

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

With the **F limited** toggle switch you can prevent the SS1 reaction for safe stopping of drives when the guard door is opened.

With the **F limited** toggle switch the control limits the speed of the axes and rotational speed of the spindle to the values defined by the machine manufacturer. The limitation depends on the active safety-related SOM_x operating mode. You can select the safety-related operating mode with the keylock switch.

> In the safety-related operating mode SOM_1, the control stops the axes and spindles when the guard door is opened.

In the **Positions** and **Status** workspaces the control shows the feed rate in orange.

Further information: "POS tab", Page 179

39.1 Checking axis positions manually

Refer to your machine manual.
This function must be adapted by your machine manufacturer.
The machine manufacturer defines the test position.

To check the position of an axis:

- Select the **Manual** operating mode
- Select **Approach test position**
- > The control displays the axes that have not been tested in the **Positions** workspace.
- Select the desired axis in the **Positions** workspace
- Press the **NC start** key
- > The axis moves to the test position.
- > After the test position has been reached, the control issues a message.
- Press the **permissive button** on the machine operating panel
- > The control displays the axis as a tested axis.

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision while approaching the test positions.

- If necessary, move to a safe position before approaching the test positions
- Watch out for possible collisions

Notes

- Machine tools with HEIDENHAIN controls may be equipped with integrated functional safety (FS) or with external safety. This chapter refers exclusively to machines with integrated functional safety (FS).
- The machine manufacturer defines the behavior of speed-controlled FS-NC axes while the guard door is open in the machine parameter **speedPosCompType** (no. 403129). The machine manufacturer can e.g. allow switching-on of the spindle and thus enable scratching of the workpiece while the guard door is open. Refer to your machine manual.

40

Settings Application

40.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Menu item
	Machine settings	■ **Machine settings** **Further information:** "Machine settings menu item", Page 2103 ■ **General information** **Further information:** "General information menu item", Page 2106 ■ **SIK** **Further information:** "SIK menu item", Page 2107 ■ **Machine times** **Further information:** "Machine times menu item", Page 2109 ■ **Set up touch probes** **Further information:** "Setting up touch probes", Page 2084 ■ **Set up wireless handwheel** **Further information:** "HR 550FS wireless handwheel", Page 2078
OS	**Operating system**	■ **Date/Time** **Further information:** "Adjust system time window", Page 2110 ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 2111 ■ **About HEROS** **Further information:** "Information on licensing and use", Page 101 ■ **SELinux** **Further information:** "SELinux security software", Page 2112 ■ **UserAdmin** **Further information:** "User management window", Page 2168 ■ **Current User** **Further information:** "Active user window", Page 2168 ■ **Touchscreen configuration** You can choose the touchscreen's sensitivity and whether touchpoints should be shown or hidden.

Icon	Category	Menu item
	Network/Remote Access	■ **Shares** **Further information:** "Network drives on the control", Page 2113 ■ **Network** **Further information:** "Ethernet interface", Page 2116 ■ **PKI Admin** Manage certificates for the control (e.g., for **OPC UA NC Server**) **Further information:** "OPC UA NC Server (options 56 to 61)", Page 2123 ■ **OPC UA** **Further information:** "OPC UA NC Server (options 56 to 61)", Page 2123 ■ **DNC** **Further information:** "DNC menu item", Page 2128 ■ **Embedded Workspace** Show the connection status **Further information:** "Embedded Workspace (option 133)", Page 2088 ■ **Printer** **Further information:** "Printers", Page 2130 ■ **VNC** **Further information:** "VNC menu item", Page 2133 ■ **Remote Desktop Manager** **Further information:** "Remote Desktop Manager window (option 133)", Page 2137 ■ **Real VNC Viewer** Define settings for external software accessing the control (e.g., for maintenance purposes); for network specialists ■ **Firewall** **Further information:** "Firewall", Page 2143

Icon	Category	Menu item
	Diagnostics/Maintenance	■ **Terminal program** Enter and execute console commands ■ **HeLogging** Define settings for internal diagnostic files ■ **Portscan** **Further information:** "Portscan", Page 2146 ■ **perf2** Check processor load and process load ■ **RemoteService** **Further information:** "Remote servicing", Page 2147 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 2148 ■ **TNCdiag** **Further information:** "TNCdiag", Page 2152 ■ **TNCscope** Software for data recording ■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 2148 ■ **Touchscreen cleaning** The control disables the touchscreen for 90 seconds. ■ **Update documentation** **Further information:** "Update documentation", Page 2150
	OEM settings	Settings for the machine manufacturer
	Machine parameters	The group contains machine parameters that can be edited, depending on your rights (e.g., **MPs for setters**). **Further information:** "Machine parameters", Page 2152
	Parameter files	Settings for the machine manufacturer
	Configurations	**Configurations** **Further information:** "Configuring the control's user interface", Page 2157
FS	Functional safety	■ **Axis status** **Further information:** "Axis status menu item", Page 2095 ■ **Safety parameters** **Further information:** "Functional safety application", Page 2094

40.2 Code numbers

Application

The top part of the **Settings** application contains the **Code number:** input field. This input field is accessible from every group.

Description of function

You can enable the following functions or areas with code numbers:

Code number	Function
123	Editing machine-specific user parameters **Further information:** "Machine parameters", Page 2152
555343	Special functions for programming with variables **Further information:** "Variable Programming", Page 1361
0	Resetting active code numbers

> The control indicates whether the caps lock key is pressed during entry. This helps to avoid incorrect entries.

40.3 Machine settings menu item

Application

In the **Machine settings** menu item of the **Settings** application, you can define the settings for simulation and program run.

Related topics

- Graphic settings for simulation

  Further information: "Simulation settings window", Page 1541

Description of function

Unit of measure area

In the **Unit of measure** area you can choose between mm and inch.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), the value is displayed to 4 decimal places

If the display in inches is active, the control also displays the feed rate in inches/min. In an inch-based program, you must multiply the feed rate by 10 before entering it.

Channel settings

The control displays the channel settings separately for the **Editor** operating mode and the **Manual** and **Program Run** operating modes.

You can define the following settings:

Setting	Meaning
Active kinematics	Use the **Active kinematics** function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, are programmed for other machines. The control offers a selection menu with all available kinematics models. The machine manufacturer defines which kinematics models you can choose. The control displays the active kinematics model in the **Machine** mode of the **Simulation** workspace.
Generate tool-usage file	The control uses the tool-usage file to check tool usage. **Further information:** "Tool usage test", Page 312 You select when the control should generate a tool-usage file: ■ **Never** The control does not generate a tool-usage file. ■ **Once** The next time you simulate or run an NC program, the control will generate a tool-usage file once. ■ **Always** When you simulate or run an NC program, the control will generate a tool-usage file each time.

Traverse limits

Use the **Traverse limits** function to limit the possible traverse path of an axis. You can define traverse limits for each axis (e.g., to protect an indexing head from collision).

The **Traverse limits** function consists of a table with the following contents:

Column	Meaning
Axis	The TNC displays each axis of the active kinematics model in a row.
Status	If you have defined one or both limits, the control displays the contents **Valid** or **Invalid**.
Lower limit	You define the lower traverse limit of the axis in this column. You can enter up to four decimal places.
Upper limit	You define the upper traverse limit of the axis in this column. You can enter up to four decimal places.

The defined traverse limits are valid across power cycles of the control, until you delete all values from the table.

The following general conditions apply to the traverse limit values:

- The lower limit must be smaller than the upper limit.
- The upper and lower limit may not both equal 0.

Other conditions apply to traverse limits for modulo axes.

Further information: "Notes on software limit switches for modulo axes", Page 1311

Notes

NOTICE

Danger of collision!

You can also select any stored kinematics model as the active machine kinematics. The control then executes all manual movements and machining operations using the selected kinematics. All subsequent axis movements pose a risk of collision!

- Use the **Active kinematics** function for the simulation only
- Use the **Active kinematics** function for selecting the active machine kinematics only if required

- In the optional machine parameter **enableSelection** (no. 205601), the machine manufacturer defines for each kinematics model whether the **Active kinematics** function can be selected.
- You can open the tool-usage file in the **Tables** operating mode.

  Further information: "Tool usage file", Page 2029
- If the control has generated a tool-usage file for an NC program, then there are contents in the **T usage order** and **Tooling list** tables (option 93).

  Further information: "T usage order (option 93)", Page 2031

  Further information: "Tooling list (option 93)", Page 2033

40.4 General information menu item

Application

In the **General information** menu item of the **Settings** application, the control provides the information about the control and the machine.

Description of function

Version information area

The control displays the following information:

Sub-area	Meaning
HEIDENHAIN	■ **Control model** Designation of the control (managed by HEIDENHAIN) ■ **NC-SW** Number of the NC software (managed by HEIDENHAIN) ■ **NCK** Number of the NC software (managed by HEIDENHAIN)
PLC	**PLC-SW** Number or name of the PLC software (managed by the machine manufacturer)

The machine manufacturer can add further software numbers (e.g., that of a connected camera).

Info about machine manufacturer area

The control shows the contents of the optional machine parameter **CfgOemInfo** (no. 131700). The control displays this area only if the machine manufacturer defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 2124

Machine information area

The control shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 2124

40.5 SIK menu item

Application

Use the **SIK** menu item of the **Settings** application to view control-specific information (e.g., the serial number and the available software options).

Related topics

- Software options on the control

  Further information: "Software options", Page 94

Description of function

SIK information area

The control displays the following information:

- **Serial number**
- **Control model**
- **Performance class**
- **Features**
- **Status**

OEM key area

In the **OEM key** area the machine manufacturer can define a manufacturer-specific password for the control.

General key area

In the **General key** area the machine manufacturer can enable all software options once for a period of 90 days (e.g., for testing).

The control indicates the status of the general key:

Status	Meaning
NONE	The general key has not yet been used for this software version.
dd.mm.yyyy	Date up to which all software options will be available. Once the general key has expired, it cannot be used again.
EXPIRED	The general key has expired for this software version.

If the software version of the control is increased (e.g., by an update), then the **General key** can be used again.

Software options area

In the **Software options** area the control shows all available software options in a table.

Column	Meaning
#	Number of the software option
Option	Name of the software option
Expiration date	The machine manufacturer can enable software options for a limited time. In this case the control shows in this column the date through which the software option is available.
	The machine manufacturer uses the **Set** button to enable a software option. For enabled software options the control displays the text **Enabled**.

40.5.1 Viewing of software options

To view enabled software options on the control:

- Select the **Home** operating mode
- Select the **Settings** application
- Select **Machine settings**
- Select **SIK**
- Navigate to the **Software options** area
- > For enabled software options the control displays the text **Enabled** at the end of the row.

Definition

Abbreviation	Definition
SIK (System Identification Key)	**SIK** is the designation of the plug-in board for the control hardware. Each control can clearly be identified by the serial number of the **SIK**.

40.6 Machine times menu item

Application

In the **Machine times** menu item of the **Settings** application, the control shows the run times since initial setup.

Related topics

- Date and time of the control

  Further information: "Adjust system time window", Page 2110

Description of function

The control displays the following machine times:

Machine time	Meaning
Control on	Run time of the control since being put into service
Machine on	Run time of the machine tool since being put into service
Program Run	Run time of all program runs since being put into service

Refer to your machine manual.
The machine manufacturer can define up to 20 additional run times.

40.7 Adjust system time window

Application

In the **Adjust system time** window you can set the time zone, date and time manually or by means of an NTP server synchronization.

Related topics

- Run times of the machine tool
  Further information: "Machine times menu item", Page 2109

Description of function

The **Date/Time** menu item opens the **Adjust system time** window. The menu item is in the **Operating system** group of the **Settings** application.

The **Adjust system time** window consists of the following areas:

Area	Function
Set the time manually	Activate this check box to define the following data: ■ Year ■ Month ■ Day ■ Time
Synchronize the time over NTP server	If you activate this check box, the control will automatically synchronize the system time with the defined NTP server. You can add a server with a host name or a URL.
Time zone	You can select your time zone from a list.

40.8 Conversational language of the control

Application

You use the **helocale** window to change the conversational language of the HEROS operating system and the machine parameters to change the NC conversational language of the control's user interface.

The HEROS conversational language only changes after a restart of the control.

Related topics

- Machine parameters of the control

  Further information: "Machine parameters", Page 2152

Description of function

You can't define two different conversational languages for the operating system and control.

The **Language/Keyboards** menu item opens the **helocale** window. The menu item is in the **Operating system** group of the **Settings** application.

The **helocale** window consists of the following areas:

Area	Function
Language	Choose the HEROS conversational language from a selection menu Only if the machine parameter **applyCfgLanguage** (no. 101305) is defined as **FALSE**.
Keyboards	Select the language layout of the keyboard for HEROS functions

40.8.1 Changing the language

By default, the control assumes the NC conversational language for the HEROS conversational language.

To change the NC conversational language:

- ▶ Select the **Settings** application
- ▶ Enter the code number 123
- ▶ Select **OK**
- ▶ Select **Machine parameters**
- ▶ Double-tap or double-click **MPs for setters**
- > The control opens the **MPs for setters** application.
- ▶ Navigate to the machine parameter **ncLanguage** (no. 101301)
- ▶ Select the desired language

- ▶ Select **Save**
- > The control opens the **Configuration data changed. All changes.** window.

- ▶ Select **Save**
- > The control opens the notification menu and displays a "Question type" error.

- ▶ Select **CLOSE CONTROL**
- > The control restarts.
- > Once the control has restarted, the NC conversational language and the HEROS conversational language are changed.

Note

Use the machine parameter **applyCfgLanguage** (no. 101305) to define whether the control assumes the setting for the NC conversational language for the HEROS conversational language.

- **TRUE** (default): The control assumes the NC conversational language. You can change the language only in the machine parameters.

  Further information: "Changing the language", Page 2112
- **FALSE**: The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.

40.9 SELinux security software

Application

SELinux is an extension for Linux-based operating systems in the sense of Mandatory Access Control (MAC). The security software protects the system against the execution of unauthorized processes or functions, i.e. viruses and other malicious software.

The machine manufacturer defines the **SELinux** settings in the **Security Policy Configuration** window.

Related topics

- Security settings with firewall

  Further information: "Firewall", Page 2143

Description of function

The **SELinux** menu item opens the **Security Policy Configuration** window. The menu item is in the **Operating system** group of the **Settings** application.

The access control of **SELinux** is regulated as follows by default:

- The control executes only programs that are installed with the HEIDENHAIN NC software.
- Only explicitly selected programs can modify safety-relevant files, such as **SELinux** system files or HEROS boot files.
- New files created by other programs may not be run.
- USB data carriers can be deselected.
- Only two processes can run new files:
  - Software update: A software update from HEIDENHAIN can replace or modify system files.
  - SELinux configuration: The configuration of **SELinux** with the **Security Policy Configuration** window is usually password-protected by the machine manufacturer (refer to the relevant machine manual).

Note

HEIDENHAIN recommends using **SELinux** as additional protection against attacks from outside the network.

Definition

Abbreviation	Definition
MAC (mandatory access control)	MAC means that the control performs only explicitly permitted actions. **SELinux** is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can be performed only if the standard functions and access control of **SELinux** permit it.

40.10 Network drives on the control

Application

Use the **Mount Setup** window to connect network drives to the control. If a network drive is connected to the control, the control displays additional drives in the navigation column of the file management.

Related topics

- File management

  Further information: "File management", Page 1144
- Network settings

  Further information: "Ethernet interface", Page 2116

Requirements

- Existing network connection
- Control and computer in same network
- Path and access data of drive to be connected are known

Description of function

The **Shares** menu item opens the **Mount Setup** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

You can also open the window with the **Mount network share** button of the **Files** operating mode.

Further information: "File management", Page 1144

You can define any number of network drives, but only seven can be connected at a time.

Network drive area

In the **Network drive** area, the control shows a list of all defined network drives, as well as the status of each drive.

The control displays the following buttons:

Button	Meaning
Mount	Connect a network drive The control selects the check box in the **Mount** column if an active connection exists.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive when the control is booting. The control selects the check box in the **Auto** column if an active automatic connection exists.
Add	Define a new connection **Further information:** "Mount assistant window", Page 2115
Remove	Delete an existing connection
Copy	Copy connection **Further information:** "Mount assistant window", Page 2115
Edit	Edit the connection settings **Further information:** "Mount assistant window", Page 2115
Private network drive	User-specific connection if user administration is active The control selects the check box in the **Privat** column if a user-specific connection exists.

Status Log area

In the **Status Log** area, the control shows status information and error messages about connections.

Use the **Clear** button to delete the contents of the **Status Log** area.

Mount assistant window

In the **Mount assistant** window you define the settings for a connection with a network drive.

The **Add**, **Copy** and **Edit** buttons open the **Mount assistant** window.

The **Mount assistant** window contains tabs with the following settings:

Tab	Setting
Drive name	■ **Drive name:** Network drive name in the file management of the control The control permits only capitals, with a colon (**:**) at the end. ■ **Private network drive** When user administration is active, the connection is only visible to the user who created it.
Share type	Transfer protocol ■ **Windows share (CIFS/SMB) or Samba server** ■ **UNIX share (NFS)**
Server and Share	■ **Server name:** Server name or IP address ■ **Share name:** Directory accessed by the control
Automount	**Connect automatically (not possible with the "Ask for password?" option)** The control connects the network drive automatically during the starting process.
User name and password (only with Windows share)	■ **Single Sign On** When user administration is active, the control automatically connects an encrypted network drive when the user logs in. ■ **Windows user name:** ■ **Ask for password? (not possible with the "Connect automatically" option)** Select whether a password is required upon connecting. ■ **Password** ■ **Password verification**
Mounting options	**Parameters for mount option "-o":** Auxiliary parameters for the connection **Further information:** "Examples of Mounting options", Page 2116
Check	The control displays a summary of the defined settings. You can check the settings and save them with **Apply**.

Examples of Mounting options

Enter options without a space, only separated by a comma

Options for SMB

Example	Meaning
domain=xxx	Name of the domain HEIDENHAIN recommends not to write the domain into the user name, but as an option.
vers=2.1	Protocol version

Options for NFS

Example	Meaning
rsize=8192	Packet size in bytes for data reception Input: **512...8192**
wsize=4096	Packet size in bytes for data transmission Input: **512...8192**
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will try to connect again
sec=ntlm	Authentication method ntlm Use this option if the control displays the **Permission denied** error message upon connecting.
nfsvers=2	Protocol version

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

40.11 Ethernet interface

Application

The control is provided with an Ethernet interface as a standard feature so that you can integrate it into a network.

Related topics

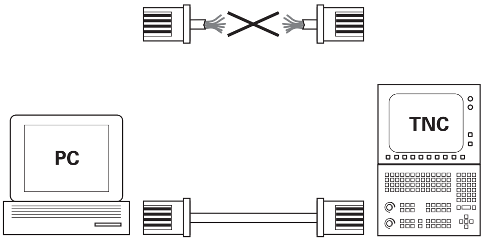
- Firewall settings
  Further information: "Firewall", Page 2143
- Network drives on the control
  Further information: "Network drives on the control", Page 2113
- External access
  Further information: "DNC menu item", Page 2128

Description of function

The control transfers data via the Ethernet interface using the following protocols:

- **CIFS** (common internet file system) or **SMB** (server message block)
  The control supports versions 2, 2.1 and 3 of these protocols.
- **NFS** (network file system)
  The control supports versions 2 and 3 of this protocol.

Connection options



You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

Use a Twisted Pair cable to connect the control to your network.

> The maximum cable length permissible between the control and a node depends on the quality grade of the cable, the sheathing, and the type of network.

Ethernet connection icon

Icon	Meaning
	Ethernet connection The control displays the icon at the bottom right in the taskbar. **Further information:** "Taskbar", Page 2188 When you click the icon, the control opens a pop-up window. The pop-up window contains the following information and functions: ■ Connected networks You can disconnect the network connection. Select the network name to reconnect. ■ Available networks ■ VPN connections Currently no function

Notes

- Protect your data and the control by running the machines in a secure network.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

40.11.1 Network settings window

Application

In the **Network settings** window you define the settings for the control's Ethernet interface.

> Have a network specialist configure the control.

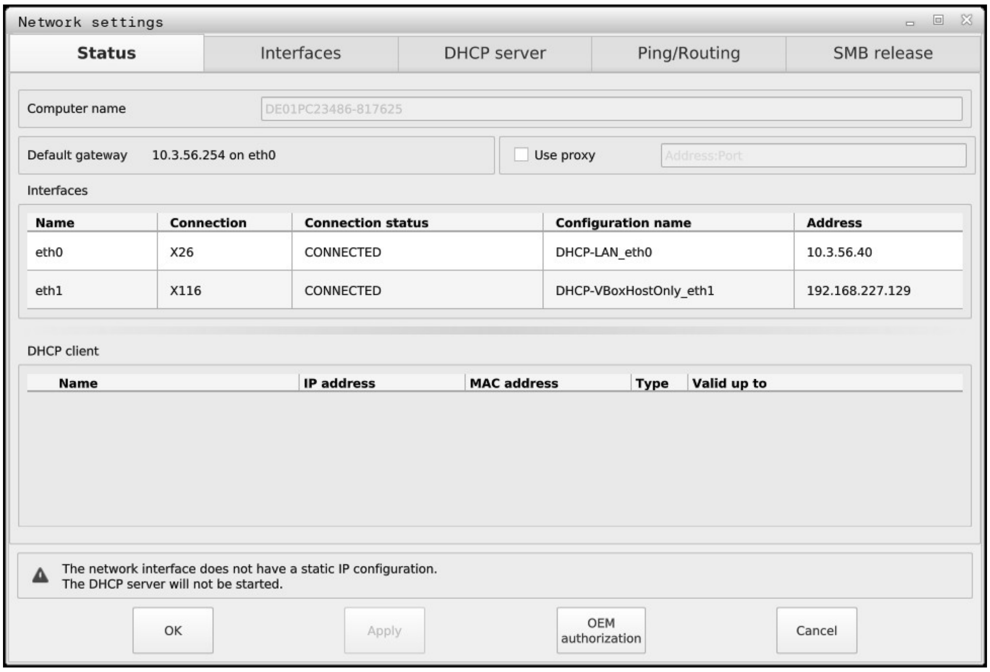
Related topics

- Network configuration
  Further information: "Network configuration with Advanced Network Configuration", Page 2196
- Firewall settings
  Further information: "Firewall", Page 2143
- Network drives on the control
  Further information: "Network drives on the control", Page 2113

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **Network**



Network settings window

Status tab

The **Status** tab contains the following information and settings:

<table>
<tr><th>Domain</th><th>Information or Setting</th></tr>
<tr><td>Computer name</td><td>The control displays the name under which the control is visible in the company network. You can change the name.</td></tr>
<tr><td>Default gateway</td><td>The control shows the default gateway and the Ethernet interface being used.</td></tr>
<tr><td>Use proxy</td><td>You can define the address and the port of a proxy server in the network.</td></tr>
<tr><td>Interfaces</td><td>The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty.
The control displays the following information in the table:
Name, e.g. eth0
Connection, e.g. X26
Connection status, e.g. CONNECTED
Configuration name, e.g. DHCP
Address, e.g. 10.7.113.10
Further information: "Interfaces tab", Page 2120</td></tr>
<tr><td>DHCP client</td><td>The control displays an overview of the devices that have received a dynamic IP address in the machine network. If there are no connections to other network components of the machine network, the table is empty.
The control displays the following information in the table:
Name
Host name and connection status of the device.
The control shows the following connection status:
Green: Connected
Red: No connection
IP address
Dynamically assigned network address of the device
MAC address
Physical address of the device
Type
Type of connection
The control displays the following connection types:
TFTP
DHCP
Valid up to
Time until which the IP address is valid without being renewed
The machine manufacturer can make settings for these devices. Refer to your machine manual.</td></tr>
</table>

Interfaces tab

The control displays the available Ethernet interfaces on the **Interfaces** tab.

The **Interfaces** tab contains the following information and settings:

Column	Information or Setting
Name	The control displays the name of the Ethernet interface. You can activate or deactivate the connection by means of a toggle switch.
Connection	The control displays the number of the network connection.
Connection status	The control displays the connection status of the Ethernet interface. The following connection statuses may be displayed: ■ **CONNECTED** Connected ■ **DISCONNECTED** Connection separated ■ **CONFIGURING** The IP address is being fetched from the server ■ **NOCARRIER** No cable present
Configuration name	You can execute the following functions: ■ Select a profile for the Ethernet interface In the factory default setting, two profiles are available: ■ **DHCP-LAN**: Settings for the standard interface for a standard company network ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network **Further information:** "Network configuration with Advanced Network Configuration", Page 2196 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 2196

The control additionally offers the following functions:

- **Set standard values**

  The control opens a pop-up window. You can import and activate profiles that you exported or that were already entered in the factory default setting.

  Further information: "Exporting and importing a network profile", Page 2122
- **Configuration name**

  You can add, edit or remove profiles for the network connection.

  > If you have changed the profile of an active connection, the control will not update the profile being used. Reconnect the corresponding interface with **Reconnect**.

  The control exclusively supports the **Ethernet** connection type.

  Further information: "Network configuration with Advanced Network Configuration", Page 2196

DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network (e.g., with industrial computers).

Refer to your machine manual.

Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

Domain	Information or Setting
Ping	**Address:Port** and **Address:** You can enter the IP address of the computer and possibly the port number for checking the network connection. Entry: Four numerical values separated by dots and, if necessary, a port number separated by a colon (e.g., **10.7.113.10:22**) As an alternative, you can enter the name of the computer whose connection you want to check. Starting and stopping the test ■ **Start** button: starts the test The control displays status information in the ping field. ■ **Stop** button: stops the test
Routing	The control displays status information of the operating system about the current routing for network administrators.

SMB share tab

The **SMB share** tab is included only in connection with a VBox programming station.

When the check box is active, the control releases areas or partitions protected by a code number for the Explorer of the Windows PC used, e.g. **PLC**. You can activate or deactivate the check box only by using the machine manufacturer code number.

In the **TNC VBox Control Panel**, select a drive letter within the **NC share** tab for displaying the selected partition and then connect the drive with **Connect**. The host displays the partitions of the programming station.

> **Further information:** Programming station for milling controls
> You download the documentation together with the programming station software.

Exporting and importing a network profile

To export a network profile:
- Open the **Network settings** window
- Select **Export configuration**
- > The control opens a window.
- Select the desired network profile
- Press **OK**
- > The control saves the network profile in the **TNC:/etc/sysconfig/net** directory.

> You can't export **DHCP** or **eth1** profiles.

To import an exported network profile:
- Open the **Network settings** window
- Select the **Interfaces** tab
- Select **Set standard values**
- > The control opens a window.
- Select **User**
- Select the desired network profile
- Press **OK**
- > The control opens a window with a prompt.
- Press **OK**
- > The control imports and activates the selected network profile.
- You might need to restart the control

Notes

- Preferably restart the control after making changes in the network settings.
- The HEROS operating system manages the **Network settings** window. You must restart the control in order to change the HEROS conversational language.

  Further information: "Conversational language of the control", Page 2111

40.12 OPC UA NC Server (options 56 to 61)

40.12.1 Fundamentals

Open Platform Communications Unified Architecture (OPC UA) describes a collection of specifications. These specifications are used to standardize machine-to-machine communication (M2M) in the field of industrial automation. OPC UA enables the data exchange across operating systems between products from different manufacturers, e.g. between a HEIDENHAIN control system and third-party software. Thus, OPC UA has become the data exchange standard for secure, reliable, manufacturer- and platform-independent industrial communication over the last years.

In 2016, the German Federal Office for Information Security (BSI) published a security analysis related to **OPC UA**. The specification analysis performed by the BSI determined that **OPC UA** provides a high level of security as compared to most other industrial protocols.

HEIDENHAIN follows the BSI recommendations and provides SignAndEncrypt, which exclusively features up-to-date IT security profiles. For this purpose, OPC UA-based industrial applications and the **OPC UA NC Server** exchange certificates for authentication. In addition, any transferred data is encrypted. This effectively prevents messages between the communication partners from being intercepted or altered.

Application

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The **OPC UA NC Server** allows you to access the data and functions of the HEIDENHAIN NC information model exposed in the server address space.

> Pay attention to the interface documentation of the **OPC UA NC Server** as well as the documentation of the client application.

Related topics

- **Information Model** interface documentation with the specification of the **OPC UA NC Server** in English

  ID: 1309365-xx or **OPC UA NC Server interface documentation**
- Quickly and easily connecting the OPC UA client application to the control

  Further information: "OPC UA connection wizard function (options 56 to 61)", Page 2126

Requirements

- OPC UA NC Server (software options 56 to 61)

  For OPC UA-based communication, the HEIDENHAIN control provides the **OPC UA NC Server**. For each OPC UA client to be connected, you need one of the six available software options (56 to 61).
- Firewall configured

  Further information: "Firewall", Page 2143
- OPC UA client supports the **Security Policy** and the authentication method of the **OPC UA NC Server**:
  - **Security Mode: SignAndEncrypt**
  - **Algorithm: Basic256Sha256**
  - **User Authentication: X509 certificates**

Description of function

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The control supports the following OPC UA functions:

- Write and read variables
- Subscribe to value changes
- Run methods
- Subscribe to events
- Read and write tool data (the corresponding right is required)
- File system access to the **TNC:** drive
- File system access to the **PLC:** drive (the corresponding right is required)

Machine parameters in conjunction with OPC UA

The **OPC UA NC Server** enables OPC UA client applications to query general machine information, such as the year of construction of the machine or its location.

The following machine parameters are available for the digital identification of your machine:

- For users: **CfgMachineInfo** (no. 131700)

  Further information: "Machine information area", Page 2106
- For the machine tool manufacturer: **CfgOemInfo** (no. 131600)

  Further information: "Info about machine manufacturer area", Page 2106

Access to directories

The **OPC UA NC Server** enables read and write access to the **TNC:** and **PLC:** drives.

The following actions are permitted:

- Creation and deletion of folders
- Reading, editing, copying, moving, creating, and deleting of files.

While the NC software is running, the files referenced in the following machine parameters are locked against write access:

- Tables referenced by the machine manufacturer in the machine parameter **CfgTablePath** (no. 102500)
- Files referenced by the machine manufacturer in the machine parameter **dataFiles** (no. 106303, branch **CfgConfigData** no. 106300)

The **OPC UA NC Server** enables access to the control even if the NC software is switched off. You can, for example, transfer automatically created service files at any time as long as the operating system is active.

NOTICE

Caution: potential damage to property!

The control does not automatically back up the files before editing or deletion. Files that are missing cannot be restored. The removal or editing of system-relevant files, such as the tool table, can negatively affect the control functions.

- System-relevant files must be edited only by authorized specialists

Required certificates

The **OPC UA NC Server** requires three different types of certificates. The server and the client need two of them, the application instance certificates, in order to establish a secure connection. The third certificate (user certificate) is required for authorization and for starting a session with specific user permissions.

The control automatically generates a two-level certificate chain referred to as the **Chain of Trust** for the server. This certificate chain consists of a self-signed root certificate (including a **revocation list**) and a certificate for the server that is created on the basis of the root certificate.

The client certificate must be added on the **Trusted** tab of the **PKI Admin** function.

All other certificates should be added on the **Issuers** tab of the **PKI Admin** function for verification of the entire certificate chain.

User certificate

The control uses the HEROS functions **Current User** or **UserAdmin** for administration of the user certificate. When you initiate a session, the rights of the associated internal user are active.

To assign a user certificate to a user:

- Open the **Current User** HEROS function
- Select **SSH keys and certificates**
- Press the **Import certificate** soft key
- > The control opens a pop-up window.
- Select the certificate
- Select **Open**
- > The control imports the certificate.
- Press the **Use for OPC UA** soft key

Self-generated certificates

You can also create and import all of the required certificates yourself.

Self-generated certificates must fulfill the following requirements:

- General requirements
  - File format: *.der
  - Signature with hash SHA256
  - Validity period of at most 5 years is recommended
- Client certificates
  - Host name of the client
  - Application URI of the client
- Server certificates
  - Host name of the control
  - Application URI of the server according to the following structure: urn:<hostname>/HEIDENHAIN/OpcUa/NC/Server
  - Validity period of 20 years maximum

Note

OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **OPC UA NC Server**.

40.12.2 OPC UA menu item (options 56 to 61)

Application

In the **OPC UA** menu item of the **Settings** application, you can set up the connections to the control and check the status of the **OPC UA NC Server**.

Description of function

Select the **OPC UA** menu item in the **Network/Remote Access** group.

The **OPC UA NC Server** area contains the following functions:

Function	Meaning
Status	Shows with an icon whether the **OPC UA NC Server** is active: ■ Green icon **OPC UA NC Server** is active ■ Gray icon: **OPC UA NC Server** is not active or software option not enabled
OPC UA connection wizard	Open the **OPC UA NC Server connection wizard** window **Further information:** "OPC UA connection wizard function (options 56 to 61)", Page 2126
OPC UA license settings	Open the **OPC UA NC Server - License Settings** window **Further information:** "OPC UA license settings function (options 56 to 61)", Page 2127
Host computer operation	Activate or deactivate host computer operation with a toggle switch **Further information:** "DNC area", Page 2128

40.12.3 OPC UA connection wizard function (options 56 to 61)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection wizard** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

- Assigning an OPC UA client to a software option 56 to 61 with the **OPC UA NC Server - License Settings** window
- Managing certificates with the **PKI Admin** menu

Description of function

The **OPC UA connection wizard** function of the **OPC UA** menu item opens the **OPC UA NC Server connection wizard** window.

Further information: "OPC UA menu item (options 56 to 61)", Page 2126

The assistant features the following steps:

- Export **OPC UA NC Server** certificates
- Import the certificates of the OPC UA client application
- Assign each of the available **OPC UA NC Server** software options to an OPC UA client application
- Import user certificates
- Assign user certificates to users
- Configure the firewall

If at least one of the options 56 to 61 is active, then, when booting for the first time, the control creates the server certificate as part of a self-generated certificate chain. The client application or the manufacturer of the application creates the client certificate. The user certificate is linked to the user account. Please contact your IT department.

Note

The **OPC UA NC Server connection wizard** also helps you create test or sample certificates for users and the OPC UA client application. Do not use the user and client application certificates created at the control for other purposes than development at the programming station.

40.12.4 OPC UA license settings function (options 56 to 61)

Application

Use the **OPC UA NC Server - License Settings** window to assign one OPC UA client application to one of the software options 56 to 61.

Related topics

- Setting up an OPC UA client application with the **OPC UA connection wizard** function

  Further information: "OPC UA connection wizard function (options 56 to 61)", Page 2126

Description of function

After using the **OPC UA connection wizard** or the **PKI Admin** menu item to import a certificate of an OPC UA client application, you can choose the certificate from a selection window.

If you activate the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

40.13 DNC menu item

Application

With the **DNC** menu item you can grant or restrict access to the control (e.g., connections over a network).

Related topics

- Connecting network drives
  Further information: "Network drives on the control", Page 2113
- Setting up a network
  Further information: "Ethernet interface", Page 2116
- TNCremo
  Further information: "PC software for data transfer", Page 2191
- Remote Desktop Manager (option 133)
  Further information: "Remote Desktop Manager window (option 133)", Page 2137

Description of function

The **DNC** area contains the following icons:

Icon	Meaning
	External access to the control is active
	Add computer-specific connection
	Edit computer-specific connection
	Delete computer-specific connection

DNC area

In the **DNC** area you use toggle switches to activate the following functions:

Switches	Meaning
DNC access permitted	Permit or block all accesses to the control through a network or a serial connection
TNCopt full access allowed	Depending on the machine, permit or block access for diagnostics or initial setup software
Host computer operation	Pass command control to an external host computer, for example to transfer data to the control; or end host computer operation If host computer operation is active, the control displays the **Host computer is active** message in the info bar. You cannot use the **Manual** and **Program Run** operating modes. You cannot activate host computer operation while running an NC program.

Secure connections for user

In the **Secure connections for user** area you activate the following functions:

Row	Meaning
Setup permitted	If the toggle switch is active, client applications can establish a secure connection for the current user.
Certificate management	In this row you open the **Certificate and keys** window. **Further information:** "SSH-secured DNC connection", Page 2178

Computer-specific connections

If the machine manufacturer has defined the optional machine parameter **CfgAccessControl** (no. 123400), then in the **Connections** area you can permit or block access for up to 32 connections defined by you.

The control shows the defined information in a table:

Column	Meaning
Name	Host name of the external computer
Description	Additional information
IP address	Network address of the external computer
Access	■ **Permit** The control permits network access without confirmation. ■ **Inquire** The control asks for confirmation upon a network access attempt. You can choose whether to permit or block the access once or always. ■ **Deny** The control does not permit any network access
Type	■ **Com1** Serial interface 1 ■ **Com2** Serial interface 2 ■ **Ethernet** Network connection
Active	If a connection is active, the control displays a green circle. If a connection is inactive, the control displays a gray circle.

Notes

- In the machine parameter **allowDisable** (no. 129202) the machine manufacturer defines whether the **Host computer operation** toggle switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

40.14 Printers

Application

You add and manage printers through the **Printer** menu item in the **Heros Printer Manager** window.

Related topics

- Using the **FN 16: F-PRINT** function for printing

  Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1382

Requirement

- PostScript-capable printer

  The control can communicate only with printers that understand PostScript emulation such as KPDL3. Some printers enable setting the PostScript emulation in the printer menu.

  Further information: "Note", Page 2133

Description of function

The **Printer** menu item opens the **Heros Printer Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

You can print the following files:

- Text files
- Graphic files
- PDF files

Further information: "File types", Page 1149

Once you have added a printer, the control shows the **PRINTER:** drive in the file management. The drive contains one folder for each defined printer.

Further information: "Creating a printer", Page 2133

There are various methods to start printing:

- Copying the file to be printed to the **PRINTER** drive

  The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job has been executed.

  You may also copy the file into the printer sub-directory if you wish to use a printer other than the default printer.

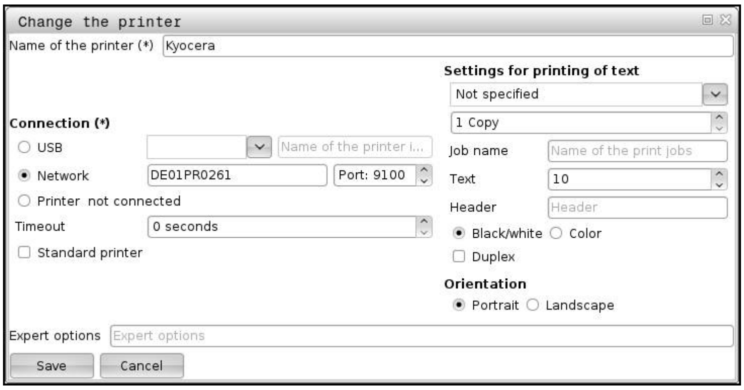
- Using the **FN 16: F-PRINT** function

Buttons

The **Heros Printer Manager** window contains the following buttons:

Button	Meaning
Create	Creates a printer
CHANGE	Adapts the properties of the selected printer
COPY	Creates a copy of the selected printer setting At first the copy has the same properties as the copied setting. This can be useful if printing both portrait and landscape formats on the same printer
DELETE	Deletes the selected printer
UP	Selects a printer
DOWN	
STATUS	Displays the status information of the selected printer
PRINT A TEST PAGE	Prints a test page on the selected printer

Change the printer window



For each printer, the following properties can be set:

Setting	Meaning
Name of the printer	Customizes the printer name
Connection	Selects the connection ■ **USB**: The control automatically displays the name ■ **Network**: Network name or IP address of the printer Port for the network printer (default: 9001) ■ **Printer %1 not connected**
Timeout	Delays the printing process The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in **PRINTER:**. Use this setting if the file to be printed is populated with FN functions (e.g., when probing).
Standard printer	Selects the default printer The control automatically assigns this setting to the first printer added.
Settings for printing of text	These settings are applicable when printing text documents: ■ Paper size ■ Number of copies ■ Job name ■ Font size ■ Header ■ Print options (black and white, color, duplex)
Orientation	Portrait or landscape for all printable files
Expert options	Available only to authorized specialists

40.14.1 Creating a printer

To create a new printer:

- ▶ Enter the printer name in the name dialog
- ▶ Select **Create**
- > The control creates a new printer.
- ▶ Press **CHANGE**
- > The control opens the **Change the printer** window.
- ▶ Define the properties
- ▶ Select **Save**
- > The control applies the settings and displays the defined printer in the list.

Note

If your printer does not permit PostScript emulation, change the printer settings if possible.

40.15 VNC menu item

Application

VNC is software that shows the screen contents of a remote computer on a local computer, and also sends keyboard actions and mouse movements of the local computer to the remote computer.

Related topics

- Firewall settings

  Further information: "Firewall", Page 2143
- Remote Desktop Manager (option 133)

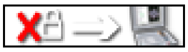
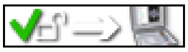
  Further information: "Remote Desktop Manager window (option 133)", Page 2137

Description of function

The **VNC** menu item opens the **VNC settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Buttons and icons

The **VNC settings** window contains the following buttons and icons:

Button and icon	Meaning
Add	Add new VNC viewer or client
Remove	Delete the selected client Only possible with manually entered clients.
Edit	Edit the configuration of the selected client
Update	Refresh view Required with connection attempts during which the dialog is open.
Set preferred owner of the focus	Activate the **Preferred owner of the focus** check box
	Another client owns the focus Mouse and keyboard are disabled
	You own the focus Entries can be made
	Prompt by another client to receive the focus Mouse and keyboard are disabled until the focus is assigned.

VNC participant settings area

In the **VNC participant settings** area, the control shows a list of all clients.

The control displays the following contents:

Column	Contents
Computer name	IP address or computer name
VNC	Connection of the client to the VNC viewer
VNC Focus	The client participates in the focus assignment
Type	■ Manual Manually entered client ■ Denied This client is not permitted to connect. ■ Enable TeleService and IPC Client via a TeleService connection ■ DHCP Other computer that retrieves an IP address from this computer.

Global settings area

In the **Global settings** area you can define the following settings:

Function	Meaning
Enable RemoteAccess and IPC	If the check box is selected, the connection is always permitted.
Password verification	Client must enter a password for verification The control opens a window when you select the check box. In this window you define the password for this client. The client must enter the password when establishing the connection.

Enabling other VNC area

In the **Enabling other VNC** area you can define the following settings:

Function	Meaning
Deny	Other VNC clients are not permitted.
Inquire	A dialog opens when another VNC client wants to connect. You must grant permission for this connection.
Permitted	Other VNC clients are permitted.

VNC Focus Settings area

In the **VNC Focus Settings** area, you can make the following settings:

Function	Meaning
Enabling VNC focus	Enables focus assignment for this system When the check box is inactive, the focus owner actively gives away the focus by using the focus symbol. The remaining clients can request the focus only after it was given away.
Reset the CapsLock key when changing the focus	When the check box is active and the focus owner has activated the CapsLock key, the CapsLock key is deactivated in a focus change. Only if the **Enabling VNC focus** check box is active
Enable Concurrency VNC Focus	When the check box is active, every client can request the focus at any time. The focus owner does not need to give away the focus before to enable that. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the **Enabling VNC focus** check box is active
Timeout Concurrency VNC Focus	Period of time after requesting the focus during which the focus owner can object to the focus change (at most 60 seconds). This period of time is set by moving a slider. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the **Enabling VNC focus** check box is active

Activate the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for purpose (e.g., ITC industrial computers).

Notes

- The machine manufacturer defines the procedure for assigning the focus with multiple clients or operating units. Focus assignment depends on the setup and operating situation of the machine tool.

  Refer to your machine manual.
- The control displays a message if the firewall settings of the control do not permit the VNC protocol for all clients.

Definition

Abbreviation	Definition
VNC (virtual network computing)	**VNC** is software with which another computer can be controlled over a network connection.

40.16 Remote Desktop Manager window (option 133)

Application

With Remote Desktop Manager you can display external computer units on the control screen that are connected via Ethernet, and operate them through the control. You can also shut down a Windows computer together with the control.

Related topics

- External access

  Further information: "DNC menu item", Page 2128

Requirement

- Remote Desktop Manager (software option 133)
- Existing network connection

  Further information: "Ethernet interface", Page 2116

Description of function

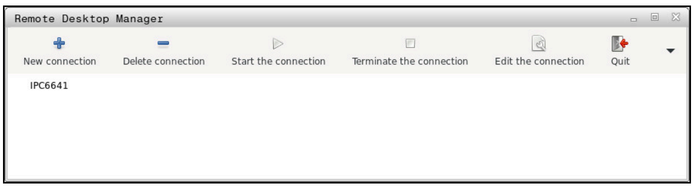
The **Remote Desktop Manager** menu item opens the **Remote Desktop Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Remote Desktop Manager grants the following connection options:

- **Windows Terminal Service (RemoteFX)**: Display the desktop of an external Windows computer on the control

  Further information: "Windows Terminal Service (RemoteFX)", Page 2138
- **VNC**: Display the desktop of an external Windows, Apple or Unix computer on the control

  Further information: "VNC", Page 2138
- **Switch-off/restart of a computer**: Automatically shut down a Windows computer together with the control
- **World Wide Web**: Only for authorized specialists
- **SSH**: Only for authorized specialists
- **XDMCP**: Only for authorized specialists
- **User-defined connection**: Only for authorized specialists



HEIDENHAIN offers the IPC 6641 as a Windows computer. With the IPC 6641 you can start and operate Windows-based applications directly from within the control.

If the desktop of the external connection or the external computer is active, all inputs from the mouse and the alphabetic keyboard are transmitted there.

When the operating system is shut down, the control automatically terminates all connections. Please note that only the connection is terminated, whereas the external computer or the external system is not shut down automatically.

Buttons

Remote Desktop Manager contains the following buttons:

Button	Function
New connection	Use the **Edit the connection** window to create a new connection **Further information:** "Establishing and starting a connection", Page 2141
Delete connection	Delete the selected connection
Start the connection	Start the selected connection **Further information:** "Establishing and starting a connection", Page 2141
Terminate the connection	Terminate the selected connection
Edit the connection	Use the **Edit the connection** window to modify the selected connection **Further information:** "Connection settings", Page 2139
Exit	Close **Remote Desktop Manager**
Import connections	Restore the selected connection **Further information:** "Exporting and importing connections", Page 2142
Export the connections	Back-up the selected connection **Further information:** "Exporting and importing connections", Page 2142

Windows Terminal Service (RemoteFX)

You don't need any additional software on a computer for a RemoteFX connection, but you might need to change some settings on the computer.

Further information: "Configuring an external computer for Windows Terminal Service (RemoteFX)", Page 2141

For integrating the IPC 6641, HEIDENHAIN recommends using a RemoteFX connection.

With RemoteFX, a separate window opens for the screen of the external computer. The active desktop on the external computer is then locked and the user logged off. This prevents two users from accessing the control simultaneously.

VNC

You need an additional **VNC** server for your external computer when connecting through VNC. Install and configure the VNC server (e.g., TightVNC server) before establishing the connection.

VNC mirrors the screen of the external computer. The active desktop on the external computer is not locked automatically.

With a **VNC** connection you can shut down the external computer through the Windows menu. The computer cannot be restarted through the connection.

Connection settings

General settings

The following settings apply to all connection options:

Setting	Meaning	Usage
Connection name	Name of the connection in **Remote Desktop Manager** You can use the following characters in the name of the connection: 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 _	Required
Restarting after end of connection	Behavior after disconnection: ■ **Always restart** ■ **Never restart** ■ **Always after an error** ■ **Ask after an error**	Required
Automatic starting upon login	Connect automatically when starting	Required
Add to favorites	The control displays the connection's icon in the taskbar. Tap or click the icon to start the connection directly.	Required
Move to the following workspace	Number of the desktop for the connection; desktops 0 and 1 are reserved for the NC software. Default setting: Third desktop	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Private connection	Connection can be seen and used only by its creator	Required
Computer	Host name or IP address of the external computer HEIDENHAIN recommends the **IPC6641.machine.net** setting for the IPC 6641. The host name **IPC6641** must be assigned to the IPC in the Windows operating system for this setting.	Required
Password	Password of the user	Required
Entries in the **Advanced options** area	Available only to authorized specialists	Optional

Additional settings for Windows Terminal Service (RemoteFX)

The control offers the following additional connection settings for the **Windows Terminal Service (RemoteFX)** option:

Setting	Meaning	Usage
User name	Name of the user	Required
Windows domain	Domain of the external computer	Optional
Full-screen mode or **User-defined window size**	Size of the connection window on the control	Required

Additional settings for VNC

The control offers the following additional connection settings for the **VNC** option:

Setting	Meaning	Usage
Full-screen mode or **User-defined window size:**	Size of the connection window on the control	Required
Permit further connections (share)	Additionally grant other VNC connections access to the VNC server	Required
View only	In display mode, the external computer cannot be operated.	Required

Additional settings for Switch-off/restart of a computer

The control offers the following additional connection settings for the **Switch-off/ restart of a computer** option:

Setting	Meaning	Usage
User name	User name with which the connection should log in.	Required
Windows domain:	If required, domain of the target computer	Optional
Max. waiting time (seconds):	A shutdown of the control causes the Windows computer to shut down as well. Before the control displays the **Now you can switch off.** message, it waits for the number of seconds defined here. While waiting, the control checks whether the Windows computer is still accessible (port 445). If the Windows computer is switched off before the defined number of seconds have expired, the control will wait no longer.	Required
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible. Windows applications may delay the shutdown of the computer after port 445 has been closed.	Required
Force	Close all programs on the Windows computer, even if dialogs are still open. If **Force** is not selected, Windows waits up to 20 seconds. This delays the shutdown process or the Windows computer is switched off before Windows has shut down.	Required
Restart	Restart the windows computer	Required
Run during restart	When the control restarts, restart the Windows computer as well. Effective only if the control is restarted using the shutdown icon at the bottom right in the taskbar or if it is restarted as a result of a change in the system settings (e.g. network settings).	Required
Run during switch-off	Shut down the Windows computer (no restart) when shutting down the control. This is the default behavior. Even the **END** key will then not trigger a restart.	Required

40.16.1 Configuring an external computer for Windows Terminal Service (RemoteFX)

To configure the external computer (e.g., in Windows 10 operating systems):

- Press the Windows key
- Select **Control Panel**
- Select **System and Security**
- Select **System**
- Select **Remote Settings**
- > The computer opens a pop-up window.
- Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
- In the **Remote Desktop** area, enable **Allow Remote connections to this computer**
- Press **OK** to confirm your settings

40.16.2 Establishing and starting a connection

To establish and start a connection:

- Open **Remote Desktop Manager**
- Select **New connection**
- > The control displays a selection menu.
- Select a connection option
- Under **Windows Terminal Service (RemoteFX)**, select the operating system
- > The control opens the **Edit the connection** window.
- Define the connection settings

  Further information: "Connection settings", Page 2139
- Press **OK**
- > The control saves the settings and closes the window.
- Select connection
- Select **Start the connection**
- > The control starts the connection.

40.16.3 Exporting and importing connections

To export a connection:

- Open **Remote Desktop Manager**
- Select the desired connection
- Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- Select **Export the connections**
- > The control opens the **Select export file** window.
- Define the name of the saved file
- Select the target file
- Select **Save**
- > The control saves the connection data under the name defined in the window.

To import a connection:

- Open **Remote Desktop Manager**
- Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- Select **Import connections**
- > The control opens the **Select file to import** window.
- Select file
- Select **Open**
- > The control creates the connection under the name that was defined originally in the **Remote Desktop Manager**.

Notes

NOTICE

Caution: Data may be lost!

If you do not shut down external computers properly, data may be irreversibly damaged or deleted.

- Configure the automatic shutdown of the Windows computer

- When you edit an existing connection, the control will automatically delete all impermissible characters from the name.

Notes in connection with the IPC 6641

- HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641. No guarantee is given for other combinations and connections.
- If you use the computer name **IPC6641.machine.net** to connect an IPC 6641, it is important to enter **.machine.net**.

  With this entry, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**; this reduces the time needed for access.

40.17 Firewall

Application

With the control you can set up a firewall for the primary network interface, and for a sandbox if needed. You can block incoming network traffic for specific senders and services.

Related topics

- Existing network connection

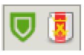
  Further information: "Ethernet interface", Page 2116
- SELinux security software

  Further information: "SELinux security software", Page 2112

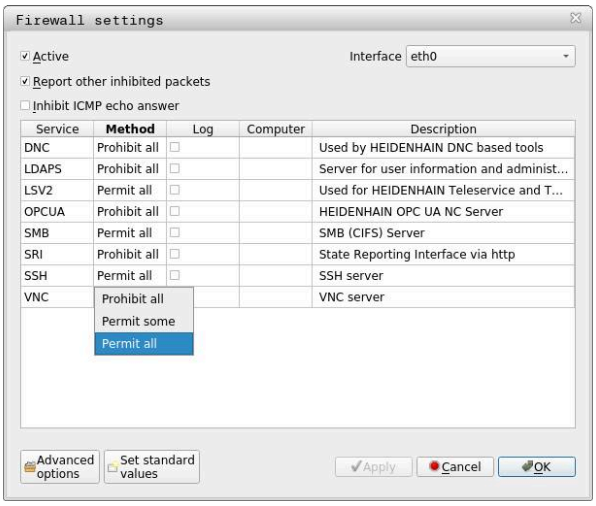
Description of function

The **Firewall** menu item opens the **Firewall settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

If you activate the firewall, the control displays an icon at the bottom right in the taskbar. The control displays the following icons, depending on the security level:

Icon	Meaning
	Firewall protection does not yet exist although it has been activated. Example: A dynamic IP address is used in the network interface configuration, but the DHCP server has not yet assigned an IP address. **Further information:** "DHCP server tab", Page 2121
	Firewall active with medium security level.
	Firewall active with high security level. All services except for SSH are blocked.

Firewall settings



The **Firewall settings** window contains the following settings:

<table>
<tr><th>Setting</th><th>Meaning</th></tr>
<tr><td>Active</td><td>Activate or deactivate firewall</td></tr>
<tr><td>Interface</td><td>Select the interface
eth0: X26 of the control
eth1: X116 of the control
brsb0: Sandbox (optional)
If a control has two Ethernet interfaces, then by default the DHCP server for the machine network is active for the second interface. With this setting you cannot activate the firewall for eth1 because the firewall and DHCP server mutually exclude each other.</td></tr>
<tr><td>Report other inhibited packets</td><td>Activate the firewall with a high security level
All services except for SSH are blocked.</td></tr>
<tr><td>Inhibit ICMP echo answer</td><td>If this check box is selected, the control does not respond to a ping request.</td></tr>
<tr><td>Service</td><td>Brief designation of services configured with the firewall. You can change the settings even if the services are not started.
DNC
DNC server using the RPC protocol for external applications that were developed with RemoTools SDK (port 19003)
For more detailed information, consult the RemoTools SDK manual.
LDAPS
Server with user data and configuration of user administration
LSV2
Functionality for TNCremo, TeleService, and other HEIDENHAIN PC tools (port 19000)
OPC UA
Service provided by the OPC UA NC Server (port 4840).
SMB
Only incoming SMB connections, meaning a Windows share on the control. Outgoing SMB connections are not influenced, meaning a Windows share connected to the control.
SSH
SecureShell protocol (port 22) for secure LSV2 handling with active user administration; starting with HEROS 504
VNC
Access to screen contents. If you block this service, then not even TeleService programs from HEIDENHAIN can access the control. If you block this service, the control displays a warning in the VNC settings window.
Further information: "VNC menu item", Page 2133</td></tr>
</table>

Setting	Meaning
Method	Configure accessibility ■ **Prohibit all**: Cannot be accessed by anyone ■ **Permit all**: Can be accessed by everyone ■ **Permit some**: Can be accessed only by specific clients In the **Computer** column you must define the computer for which access is permitted. If you do not define a computer, the control activates **Prohibit all**.
Log	The control shows the following messages when transmitting network packets: ■ Red: Network packet blocked ■ Blue: Network packet accepted
Computer	IP address or host name of the computers with access rights. Separated by commas, if there are multiple computers The control converts the host name to an IP address when the control starts. If the IP address changes, you must restart the control or change the setting. The control issues an error message if it cannot convert the host name to an IP address. Only with the **Permit some** method
Advanced options	Only for network specialists
Set standard values	Reset the settings to the default values recommended by HEIDENHAIN

Notes

- Have your network specialist check and, if necessary, change the standard settings.
- 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 identification.
- The firewall does not protect the second network interface **eth1**. Connect only trustworthy hardware to this interface, and do not use this interface for Internet connections.

40.18 Portscan

Application

With the **Portscan** function, the control checks all open, incoming TCP and UDP listen ports at defined intervals or when commanded. The control shows a message if a port is not listed.

Related topics

- Firewall settings

  Further information: "Firewall", Page 2143
- Network settings

  Further information: "Network configuration with Advanced Network Configuration", Page 2196

Description of function

The **Portscan** menu item opens the **Portscan** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

The control searches for all open, incoming TCP and UDP listen ports on the system and compares them to the following whitelists:

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

Each whitelist contains the following information:

- Port type (TCP/UDP)
- Port number
- Offering program
- Comments (optional)

In the **Manual Execution** area, use the **Start** button to start the portscan manually. Use the **Automatic update on** function of the **Automatic Execution** to have the control automatically perform the portscan at a defined interval. You define the interval with a slider.

If the control performs the portscan automatically, then only ports listed in the whitelists my be open. The control shows a message window if a port is not listed.

40.19 Remote servicing

Application

In conjunction with the remote service setup tool, TeleService from HEIDENHAIN offers the possibility to establish encrypted end-to-end connections between a computer and a machine over the Internet.

Related topics

- External access

  Further information: "DNC menu item", Page 2128
- Firewall

  Further information: "Firewall", Page 2143

Requirements

- Existing Internet connection

  Further information: "Network configuration with Advanced Network Configuration", Page 2196
- **LSV2** connection permitted by the firewall

  Remote diagnosis via the TeleService PC software uses the **LSV2** service. By default, the control's firewall blocks all incoming and outgoing connections. For this reason you must permit a connection for this service.

  You have the following options for permitting the connection:

  - Deactivate the firewall
  - Define the **Permit some** method for the **LSV2** service, and enter the name of the computer under **Computer**

  Further information: "Firewall", Page 2143

Description of function

The **Remote Service** menu item opens the **HEIDENHAIN remote maintenance** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You need a valid session certificate for the servicing session.

Session certificate

During installation of the NC software, a temporary certificate is automatically installed on the control. A new installation or update may only be carried out by a service technician from the machine manufacturer.

A new certificate must be installed if no valid session certificate is installed on the control. Clarify with the service technician which certificate is needed. The service technician will then provide you with a valid certificate file, if necessary, which you must then install.

Further information: "Installing a session certificate", Page 2148

In order to start the servicing session, you must enter the session key from the machine manufacturer.

40.19.1 Installing a session certificate

To install a session certificate on the control:

- Select the **Settings** application
- Select **Network/Remote Access**
- Double-tap or double-click **Network**
- > The control opens the **Network settings** window.
- Select the **Internet** tab

> The machine manufacturer defines the settings in the **Telemaintenance** field.

- Select **Add**
- > The control displays a selection menu.
- Select file
- Select **Open**
- > The control opens the certificate.
- Select **OK**
- Restart the control to load the settings

Notes

- If you've deactivated the firewall, you must reactivate it after the servicing session!
- If you permit the **LSV2** service in the firewall, the access security is ensured through the network settings. The network security is the responsibility of the machine manufacturer or the respective network administrator.

40.20 Backup and restore

Application

With the **NC/PLC Backup** and **NC/PLC Restore** functions you can back up and restore individual folders or the entire **TNC:** drive. You can save the backup files to various types of memory media.

Related topics

- File management, **TNC:** drive

  Further information: "File management", Page 1144

Description of function

You open the backup function through the **NC/PLC Backup** menu item. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You open the restore function through the **NC/PLC Backup** menu item.

The backup function creates a ***.tncbck** file. The restore function can restore these files as well as files from existing TNCbackup programs. If you double tap or click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: "File management", Page 1144

Within the backup function you can chose between the following types of backups:

- **Back up the “TNC:” partition**

  Back-up all data on the **TNC:** drive
- **Back up the directory tree**

  Back-up the selected folders and their subfolders on the **TNC:** drive
- **Back up the machine configuration**

  Only for the machine manufacturer
- **Complete backup (TNC: and machine configuration)**

  Only for the machine manufacturer

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** buttons.

40.20.1 Backing up data

To back-up the data of the **TNC:** drive:

- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Backup**
- > The control opens the **Back up the “TNC:” partition** window.
- ▶ Specify the type of backup
- ▶ Select **Forward**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select any predefined exclusion rules or ones you have defined yourself
- ▶ Select **Forward**
- > The control generates a list of files for backing up.
- ▶ Check list
- ▶ Deselect files if necessary
- ▶ Select **Forward**
- ▶ Enter the name of the backup file
- ▶ Select the storage path
- ▶ Select **Forward**
- > The control generates the backup file.
- ▶ Confirm with **OK**
- > The control concludes the backup process and restarts the NC software.

40.20.2 Restoring data

NOTICE

Caution: Data may be lost!

When you restore data (Restore function), any existing data will be overwritten without a confirmation prompt. Existing data is not automatically backed up by the control before running the restore process. Power failures or other problems can interfere with the data restore process. As a consequence, data may be irreversibly damaged or deleted.

- Before starting the data restore process, make a backup of the existing data

To restore data:

- Select the **Settings** application
- Select **Diagnostics/Maintenance**
- Double-tap or double-click **NC/PLC Restore**
- > The control opens the **Restore data - %1** window.
- Select the archive to be restored
- Select **Forward**
- > The control generates a list of files for restoring.
- Check list
- Deselect files if necessary
- Select **Forward**
- If necessary, pause the control with **Stop NC software**
- Select **Extract archive**
- > The control restores the files.
- Confirm with **OK**
- > The control restarts the NC software.

Note

The TNCbackup PC program can also process ***.tncbck** files. TNCbackup is part of TNCremo.

40.21 Update documentation

Application

You can use the **Update documentation** function to install or update, for example, the **TNCguide** integrated product aid.

Related topics

- **TNCguide** integrated product aid

  Further information: "User's Manual as integrated product aid: TNCguide", Page 82
- Product aid on the HEIDENHAIN website

  TNCguide

Description of function

Settings ▶ Diagnostics/Maintenance ▶ Update documentation

In the **Update documentation** area, the control shows the file manager. You can select and install the desired documentation from the file manager.

Further information: "Transferring TNCguide", Page 2151

The control shows all available documentation in the **Help** application.

Further information: "Help workspace", Page 1506

> In the **Update documentation** area you can install all HEIDENHAIN-specific documentation (e.g., NC error messages).

40.21.1 Transferring TNCguide

You can find and transfer the desired **TNCguide** version as follows:

- Select the link to the HEIDENHAIN website **TNCguide**
- Select **TNC Controls**
- Select **TNC7 Series**
- Select the NC software number
- Navigate to **Product help (HTML files)**
- Select **TNCguide** in the desired language
- Select path to save the file
- Select **store**
- > The download begins.
- Transfer the downloaded file to the TNC control
  - Select the **Home** operating mode
  - Select the **Settings** application
  - Select **Diagnostics/Maintenance**
  - Select **Update documentation**
  - > The control opens the **Select installation file** area.
  - Select the desired file with extension ***.tncdoc**
  - Open: Select **Open**
  - > A pop-up window appears, stating whether the installation was successful or failed.
  - Select the **Help** application
  - Select **home**
  - > The control shows all available documentation.

40.22 TNCdiag

Application

The control displays status and diagnostic information of HEIDENHAIN components in the **TNCdiag** window.

Description of function

⚙	Only use this function after consultation with your machine manufacturer.

📖	For more information, please refer to the **TNCdiag** documentation.

40.23 Machine parameters

Application

You can configure the behavior of the control with machine parameters. The control offers the applications **MPs for users** and **MPs for setters** for this. You can open the **MPs for users** application at any time without having to enter a code number.

The machine manufacturer defines which machine parameters are in which applications. HEIDENHAIN offers a standard scope of parameters for the **MPs for setters** application. The following contents describe only the standard scope of the **MPs for setters** application.

Related topics

- List of machine parameters in the **MPs for setters** application

  Further information: "Machine parameters", Page 2202

Requirements

- Code number 123

  Further information: "Code numbers", Page 2103
- Contents of the **MPs for setters** application are defined by the machine manufacturer

Description of function

The **MPs for setters** menu item opens the **MPs for setters** application. The menu item is in the **Machine parameters** group of the **Settings** application.

In the **Machine parameters** group the control shows only those menu items that you can choose with the current access rights.

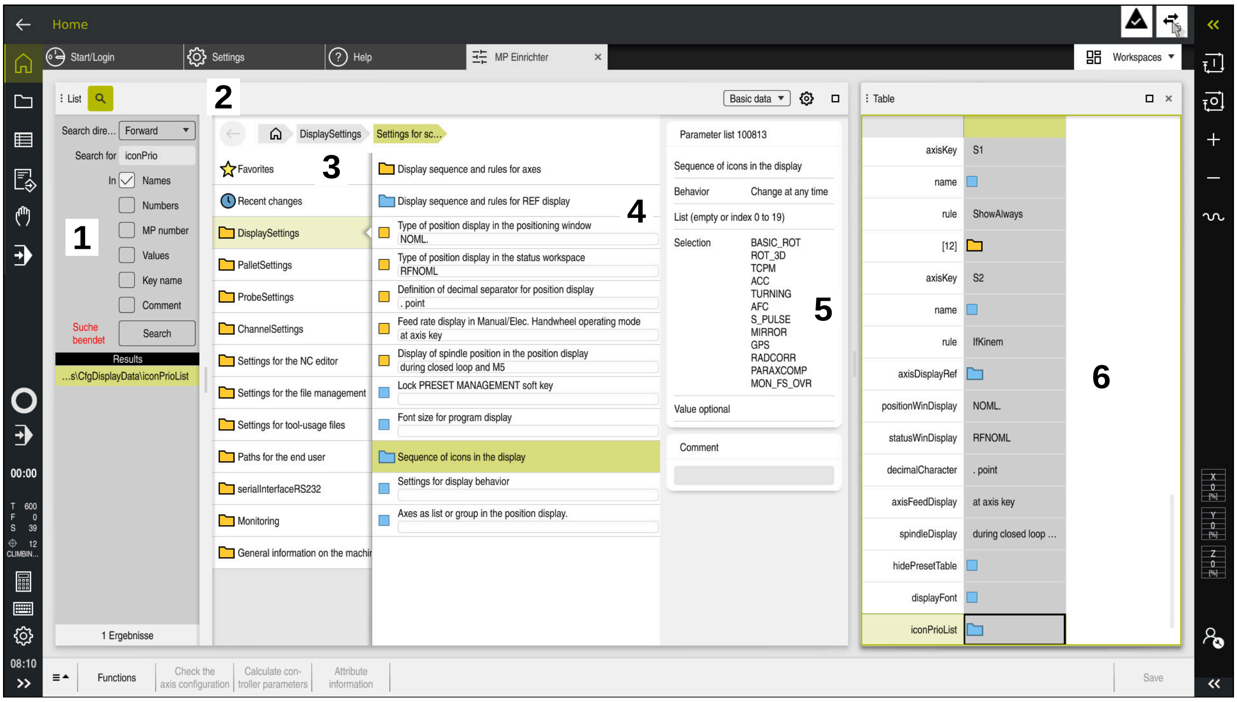
If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- **List**
- **Table**

You can't close the **List** workspace.

Areas of the configuration editor



MPs for setters application with selected machine parameter

The configuration editor shows the following areas:

1 **Search** column

You can search forward or backward with the following characteristics:

- Name

  This is the language-neutral name used for machine parameters in the User's Manual.
- Number

  This is the unique number used for machine parameters in the User's Manual.
- MP number of the iTNC 530
- Value
- Key name

  Machine parameters for axes or channels exist more than once. In order to avoid ambiguity, each axis and each channel is identified with a key name (e.g., **X1**)
- Comment

The control displays the results.

2 Title bar of the **List** workspace

You can show and hide the **Search** column, use a selection menu to filter the contents, and open the **Configuration** window.

Further information: "Configuration window", Page 2156

3 Navigation column

The control provides the following options for navigation:

- Navigation path
- Favorites
- 21 most recent changes
- Structure of the machine parameters

4 Content column

In the content column the control displays objects, machine parameters, or changes that you select using the search function or navigation column.

5 Information area

The control displays information about the selected machine parameter or change.

Further information: "Information area", Page 2156

6 **Table** workspace

In the **Table** workspace the control shows the selected contents within the structure. In order to do so, in the **Configuration** window the **Synchronized navigation in list and table** toggle switch must be set to active.

The control displays the following information:

- Name of the objects
- Icon of the objects
- Value of the machine parameters

Icons and buttons

The configuration editor contains the following icons and buttons:

Icon or button	Meaning
	Open the **Configuration** window **Further information:** "Configuration window", Page 2156
	Select **Recent changes**
	Object exists ■ Data object ■ Directory ■ Parameter list
	Object empty
	Machine parameter exists
	Optional machine parameter does not exist
	Machine parameter invalid
	Machine parameter readable but not editable
	Machine parameter unreadable and uneditable
	Changes to the machine parameter not yet saved
Functions	Open the context menu **Further information:** "Context menu", Page 1522
Check the axis configuration	Only for the machine manufacturer
Calculate controller parameters	Only for the machine manufacturer
Attribute information	Only for the machine manufacturer
Save	The control opens a window with all of the changes since the most recent saving. You can save or discard the changes.

Configuration window

In the **Configuration** window you define the settings for displaying the machine parameters in the configuration editor.

The **Configuration** window consists of the following areas:

- **List**
- **Table**

The **List** area contains the following settings:

Setting	Meaning
Show MP descriptive texts	If the toggle switch is active, the control displays a description of the machine parameter in the active conversational language. If the toggle switch is not active, the control displays the language-neutral name of the machine parameter.
Show details	Hide or show the information area with this toggle switch.

The **Table** area contains the following settings:

Setting	Meaning
Show details when table displayed	If the toggle switch is active, the control shows the information area even if the **Table** workspace is open. If the toggle switch is not active, the control shows the information area only if the **Table** workspace is closed.
Synchronized navigation in list and table	If the toggle switch is active, the control always shows in the **Table** workspace the object that is marked in the **List** workspace, and vice versa. If the toggle switch is not active, the contents of the two workspaces do not synchronize.

Information area

If you select contents from the favorites or the structure, the control will display some of the information below in the information area:

- Type of object, such as data object list or parameter, perhaps with number
- Descriptive text of machine parameter
- Information about the effect
- Permitted or required input
- Behavior, such as program run disabled
- MP number of the iTNC 530 for machine parameters
- Machine parameter optional

If you select content from one of the recent changes, the control will display the information below in the information area:

- Sequential number of the change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Information about the effect

40.24 Configuring the control's user interface

Application

Each user can create and activate configurations in which the control's user interface is individually adapted.

Related topics

- Workspaces
  Further information: "Workspaces", Page 112
- Control interface
  Further information: "Areas of the control's user interface", Page 109

Description of function

A configuration contains all adaptations to the control's user interface that do not influence the control's actual functions.

- Settings for the TNC bar
- Arrangement of workspaces
- Font size
- Favorites

You manage the configurations in the **Settings** application.

To navigate to this function:

Settings ▶ **Configurations** ▶ **Configurations**

The **Configurations** area contains the following functions:

Function	Meaning
Active configuration	Activate a configuration from a selection menu **Further information:** "Desktop menu workspace", Page 125
Default configuration	Use the **Reset** button to apply the settings of the **OEM configuration** to the active configuration.
Save as OEM configuration	The machine manufacturer uses the **Save** button to overwrite the **OEM configuration**.

The control displays the following information about all available configurations in a table:

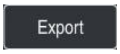
Column	Meaning
Configuration name	Name of the configuration
Selectable	If this toggle switch is active, you can select the configuration in the **Active configuration** selection menu.
Exportable	If this toggle switch is active, you can export the configuration. **Further information:** "Exporting and importing configurations", Page 2158
Edit	This column contains two buttons, for renaming and deleting the configuration.

Press the **Add** button to create a new configuration.

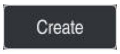
40.24.1 Exporting and importing configurations

To export configurations:

- Select the **Settings** application
- Select **Configurations**
- \> The control opens the **Configurations** area.
- Activate the **Exportable** toggle switch for the desired configuration, if necessary



- Select **Export**
- \> The control opens the **Save as** window.
- Select the target file
- Enter a file name



- Select **Create**
- \> The control saves the configuration file.

To import configurations:

- Select **Import**
- \> The control opens the **Import configurations** window.
- Select file

- Select **Import configuration**
- \> If importing a configuration would overwrite a file with the same name, the control displays a prompt.
- Select the procedure:
  - **Overwrite**: the control overwrites the original configuration.
  - **Keep**: the control does not import the configuration.
  - **Cancel**: the control cancels the import process.

Notes

- Delete only inactive configurations. If you delete an active configuration, the control first actives a default configuration. This can lead to delays.
- The **Overwrite** function permanently replaces existing configurations.

41

User administration

41.1 Fundamentals

Application

User administration enables you to create and administrate different users with different access rights to various functions of the control. You can assign roles to the various users that reflect their respective tasks, such as machine operator or setup technician.

User administration is inactive in the control's factory default setting. This status is called **legacy mode**.

Description of function

User administration supports you in the following fields of security, based on the requirements of the IEC 62443 series of standards:

- Application security
- Network security
- Platform security

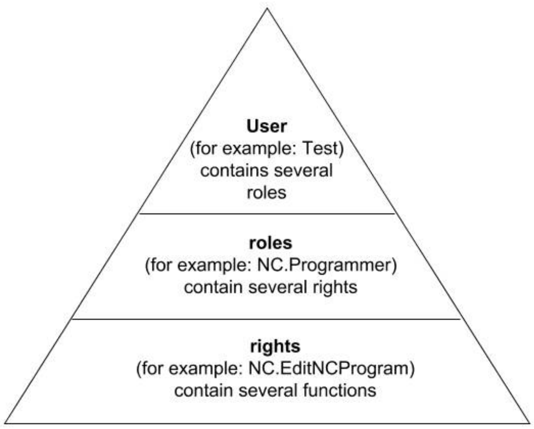
The user administration differentiates between the following terms:

- User

  Further information: "Users", Page 2160
- Roles

  Further information: "Roles", Page 2162
- Rights

  Further information: "Rights", Page 2162



Users

The user administration offers the following types of users:

- Function users pre-defined by HEIDENHAIN
- Function users pre-defined by the machine manufacturer
- Self-defined users

Depending on the task assigned, you can use one of the pre-defined function users or you have to create a new user.

Further information: "Creating a new user", Page 2166

If you deactivate user administration, the control saves all configured users. Thus they will be available again when user administration is reactivated.

If you want to delete the configured users upon deactivation, you need to set this explicitly when deactivating user administration.

Further information: "Deactivating user administration", Page 2167

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created upon activation of user administration. Function users cannot be changed.

HEIDENHAIN provides four different function users in the control's factory default setting.

- **useradmin**

  The **useradmin** function user is automatically created upon activation of user administration. The **useradmin** function user allows you to configure and edit user administration.
- **sys**

  The **sys** function user allows you to access the **SYS:** drive of the control. This function user is reserved for use by HEIDENHAIN service personnel.
- **user**

  In **legacy mode**, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect. The logged-on user of the type **user** cannot be changed in **legacy mode**.
- **oem**

  The **oem** function user is intended for the machine manufacturer. The **oem** function user allows you to access the **PLC:** drive of the control.

The useradmin function user

The **useradmin** user is comparable to the local administrator of a Windows system.

The **useradmin** account provides the following functions:

- Creating databases
- Assigning the password data
- Activating the LDAP database
- Exporting LDAP server configuration files
- Importing LDAP server configuration files
- Emergency access if the user database was destroyed
- Retroactive change of the database connection
- Deactivating user administration

Function users pre-defined by the machine manufacturer

Your machine manufacturer defines function users who are required for specific tasks such as machine maintenance.

By entering code numbers or passwords that replace code numbers, you can temporarily enable rights of **oem** function users.

Further information: "Active user window", Page 2168

The machine manufacturer's function users can already be active in **legacy mode** and replace code numbers.

Roles

HEIDENHAIN combines several rights for individual task areas to roles. Different pre-defined roles that you can use to assign rights to your users are available. The tables below describe the individual rights of the different roles.

Further information: "List of roles", Page 2259

Advantages of classification in roles:

- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

User administration offers roles for the following tasks:

- **Operating system roles:** access to functions of the operating system and interfaces
- **NC operator roles:** access to functions for programming, setting up and running NC programs
- **Machine tool builder (PLC) roles:** access to functions for configuring and checking the control

Every user should have at least one role from the operating system area and at least one role from the programming area.

HEIDENHAIN recommends permitting more than one person to access an account with the HEROS.Admin role. This ensures that necessary changes to user administration can also be made in the administrator's absence.

Local or remote registration

You can enable a role either for local login or for remote login. With local login, the user directly logs on to the control at the control's screen. A remote login (DNC) is a connection via SSH.

Further information: "SSH-secured DNC connection", Page 2178

If a role is only enabled for local login, Local. is added to the role name (e.g., Local.HEROS.Admin instead of HEROS.Admin).

If a role is only enabled for remote login, Remote. is added to the role name (e.g., Remote.HEROS.Admin instead of HEROS.Admin).

You can therefore also make the rights of a user dependent on the access used to operate the control.

Rights

The user administration is based on the Unix rights management. Access to the control is controlled by means of rights.

Rights gather various functions of the control (e.g., editing the tool table).

User administration offers rights for the following tasks:

- HEROS rights
- NC rights
- PLC rights (machine manufacturer)

If more than one role is assigned to a user, he will be granted all rights contained in these roles.

> Ensure that every user is assigned all access rights he needs. The access rights result from the tasks a user performs on the control.

The access rights of HEIDENHAIN function users are already pre-defined in the control's factory default setting.

Further information: "List of rights", Page 2263

Password settings

If you use an LDAP database, users with the HEROS.Admin role can define password requirements. For this, the control provides the **Password settings** tab.

Further information: "Saving user data", Page 2170

The following parameters are available:

Password lifetime

- **Validity period of password:**
  Here, you can indicate how long the password can be used.
- **Warning before expiration:**
  From the defined time, a warning will be issued that the password will soon expire.

Password quality

- **Minimum password length:**
  Here, you can indicate the minimum password length.
- **Minimal number of character classes (upper/lower, digits, special):**
  Here, you can indicate the minimum number of different character classes required in the password.
- **Maximum number of repeated characters:**
  Here, you can indicate the maximum number of identical successive characters in the password.
- **Maximum length of character sequences:**
  Here, you can indicate the maximum length of the character sequences to be used in the password, e.g. 123.
- **Dictionary check (number of matching characters):**
  Here, you can enable a check whether the password contains known words and specify the allowed number of meaningful characters.
- **Minimum number of characters changed compared to previous password:**
  Here, you can specify how many characters in the new password must be different from the previous one.

You define the values for each parameter on a scale.

For reasons of security, passwords should comply with the following criteria:

- Eight characters minimum
- Letters, numbers, and special characters
- Avoid using whole words or a sequence of characters (e.g., Anna or 123)

> If you want to use special characters, pay attention to the keyboard layout. HEROS assumes a US keyboard, the NC software assumes a HEIDENHAIN keyboard. External keyboards can be freely configured.

Additional directories

HOME: drive

When user administration is active, a private **HOME:** directory, to which you can save your private programs and files, is available to every user.

The **HOME:** directory can be viewed by any logged-on user.

public directory

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

The **public** directory can be accessed by any user.

In the **public** directory you can, for example, make files available to other users.

Further information: "File management", Page 1144

41.1.1 Configuring user administration

User administration needs to be configured before you can use it.

Perform the following steps for configuration:

1. Opening the **User management** window
2. Activating user administration
3. Defining the password for the **useradmin** function user
4. Setting up a database
5. Creating a new user

- You can exit the **User management** window after each configuration step.
- If you exit the **User management** window directly after having activated user administration, the control will prompt you for a restart once.

Opening the User management window

To open the **User management** window:

- Select the **Settings** application
- Select **Operating system**
- Double-tap or double-click **CurrentUser**
- > The control opens the **User management** window in the **Settings** tab.

Further information: "User management window", Page 2168

Activating user administration

To activate user administration:

- ▶ Select **User administration active**
- > The control shows the message **Password for user 'useradmin' missing**.
- ▶ Retain or reactivate the active status of the **Anonymize users in login data** function

> - The purpose of the **Anonymize users in login data** function is data privacy; this function is active by default. While this function is active, user data in all log files of the control will be anonymized.
> - If you exit the **User management** window directly after having activated user administration, the control will prompt you for a restart once.

Define the password for the useradmin function user

If you are activating user administration for the first time, you must define a password for the **useradmin** function user.

Further information: "Users", Page 2160

To define a password for the **useradmin** function user:

- ▶ Select **Password for useradmin**
- > The control opens the **Password for user 'useradmin'** pop-up window.
- ▶ Enter the password for the **useradmin** function user

> Please observe the recommendations for passwords.
> **Further information:** "Password settings", Page 2163

- ▶ Repeat the password
- ▶ Select **Set new password**
- > The control shows the message **Settings and password for 'useradmin' were changed**.

Setting up a database

To set up a database:

- ▶ Select a database for saving the user data, such as **Local LDAP database**
- ▶ Select **Configuration**
- > The control opens a window for configuring the corresponding database.
- ▶ Follow the instructions from the control in the window
- ▶ Select **APPLY**

> The following options are available for saving your user data:
> - **Local LDAP database**
> - **LDAP on remote computer**
> - **Connection to Windows domain**
>
> Parallel operation of Windows users and users from an LDAP database is possible.
> **Further information:** "Saving user data", Page 2170

Creating a new user

To create a new user:

- Select the **User management** tab
- Select **Create new user**
- > The control adds a new user to the **User list**.
- Change the name as needed
- Edit a password as needed
- Define a profile image as needed
- Enter a description as needed
- Select **Add role**
- > The control opens the **Add role** window.
- Select a role
- Select **Add**

> You can also add roles using the **Add external login** and **Add local login** buttons.
>
> **Further information:** "Roles", Page 2162

- Select **Close**
- > The control closes the **Add role** window.
- Select **OK**
- Select **APPLY**
- > The control adopts the changes.
- Select **END**
- > The control opens the **System reboot required** window.
- Select **Yes**
- > The control restarts.

> The user must change the password when logging in for the first time.

41.1.2 Deactivating user administration

User administration can only be deactivated by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "Users", Page 2160

To deactivate user administration:

- Log in as a function user
- Opening the **User management** window
- Select **User administration inactive**
- If desired, check **Delete existing user databases** to delete all configured users and user-specific directories
- Select **APPLY**
- Select **END**
- > The control opens the **System reboot required** window.
- Select **Yes**
- > The control restarts.

Notes

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in login data** function, the system will show personalized user data in all control log files.
If servicing becomes necessary or if the log files need to be transmitted for another reason, the contracting party will be able to view this user data. In this case it is your responsibility to ensure that all required data protection provisions have been made at your company.

- Retain or reactivate the active status of the **Anonymize users in login data** function

- Some user administration areas are configured by the machine manufacturer. Refer to your machine manual.
- HEIDENHAIN recommends activating user administration as part of an IT safety concept.
- If both user administration and a screensaver are active, then the current user's password must be entered to unlock the screen.

  Further information: "HEROS menu", Page 2184
- If you used **Remote Desktop Manager** to establish private connections before user administration was activated, these connections are no longer available after the activation of user administration. Save your private connections before activating user administration.

  Further information: "Remote Desktop Manager window (option 133)", Page 2137

41.2 User management window

Application

In the **User management** window you can activate and deactivate user administration, as well as define its settings.

Related topics

- **Active user** window
  Further information: "Active user window", Page 2168

Requirement

- If user administration is active, the HEROS.Admin role
  Further information: "List of roles", Page 2259

Description of function

To navigate to this function:

Settings ▶ **Operating system** ▶ **UserAdmin**

The **User management** window contains the following tabs:

Tab	Meaning
Settings	Configure user administration **Further information:** "Configuring user administration", Page 2164
User management	Create or remove users, change rights, add profile images **Further information:** "Creating a new user", Page 2166
Password settings	Define password requirements **Further information:** "Password settings", Page 2163
User-defined roles	Roles created for a Windows domain **Further information:** "Connection to Windows domain", Page 2172

41.3 Active user window

Application

In the **Active user** window, the control displays information about the logged on user, such as assigned rights. You can also manage other user settings, such as keys for SSH-secured DNC connections or smartcards for logon, and change the password.

Related topics

- SSH-secured DNC connections
  Further information: "SSH-secured DNC connection", Page 2178
- Logon with smartcards
  Further information: "Logon with smartcards", Page 2176
- Available roles and rights
  Further information: "User administration roles and rights", Page 2259

Description of function

To navigate to this function:

Settings ▶ **Operating system** ▶ **Current User**

By default the **Active user** window is on the **Base rights** tab. On this tab the control displays information about the user and all assigned rights.

When you open the **Active user** window, by default the window shows the **Base rights** tab. On this tab the control displays information about the user and all assigned rights.

The **Base rights** tab contains the following buttons:

Button	Meaning
Add rights	On the **Added rights** tab, enable rights from another user or function user until the next logoff
Open user administration	Open the **User management** window **Further information:** "User management window", Page 2168
SSH keys and certificates	Manage keys and certificates for client connections **Further information:** "SSH-secured DNC connection", Page 2178 **Further information:** "OPC UA NC Server (options 56 to 61)", Page 2123
Create token **Delete token**	Manage smartcards for logon with a card reader **Further information:** "Logon with smartcards", Page 2176
Close	Close the **Active user** window

On the **Change password** tab you can check your password against the current requirements or set a new password.

Further information: "Password settings", Page 2163

Note

In legacy mode, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect.

Further information: "Users", Page 2160

41.4 Saving user data

41.4.1 Overview

The following options are available for saving your user data:

- **Local LDAP database**

  Further information: "Local LDAP database", Page 2170
- **LDAP on remote computer**

  Further information: "LDAP database on a remote computer", Page 2171
- **Connection to Windows domain**

  Further information: "Connection to Windows domain", Page 2172

> Parallel operation of Windows users and users from an LDAP database is possible.

41.4.2 Local LDAP database

Application

With the **Local LDAP database** setting the control saves the user data locally. That way you can activate user administration even on machines without a network connection.

Related topics

- Using an LDAP database on multiple controls

  Further information: "LDAP database on a remote computer", Page 2171
- Connecting a Windows domain with user administration

  Further information: "Connection to Windows domain", Page 2172

Requirements

- User administration is active

  Further information: "Activating user administration", Page 2165
- **useradmin** user is logged on

  Further information: "Users", Page 2160

Description of function

A local LDAP database offers the following options:

- Using user administration on one single control
- Setting up a central LDAP server for more than one control
- Exporting an LDAP server configuration file if the exported database is to be used by more than one control

Setting up a Local LDAP database

To set up a **Local LDAP database**:

- Opening the **User management** window
- Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- Select **Local LDAP database**
- Select **Configuration**
- > The control opens the **Configure local LDAP database** window.
- Enter the name of the **LDAP domain**
- Enter the password
- Repeat the password
- Select **OK**
- > The control closes the **Configure local LDAP database** window.

Notes

- Before you can start editing the user administration, the control prompts you to enter the password of your local LDAP database.

  Passwords must not be trivial and must be known only to the administrators.
- If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

41.4.3 LDAP database on a remote computer

Application

With the **LDAP on remote computer** function you can transmit the configuration of a local LDAP database between controls and computers. That way you can use the same users on multiple controls.

Related topics

- Configuring an LDAP database on a control

  Further information: "Local LDAP database", Page 2170
- Connecting a Windows domain with user administration

  Further information: "Connection to Windows domain", Page 2172

Requirements

- User administration is active

  Further information: "Activating user administration", Page 2165
- **useradmin** user is logged on

  Further information: "Users", Page 2160
- LDAP database has been set up in the company network
- Server configuration file of an existing LDAP database is stored on the control or a PC in the network

  If the configuration file is stored on a PC, the PC must be running and accessible through the network.

  Further information: "Providing a server configuration file", Page 2172

Description of function

The **useradmin** function user can export the server configuration file of an LDAP database.

Providing a server configuration file

To provide a server configuration file:

- ▶ Opening the **User management** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **Local LDAP database**
- ▶ Select **Export server configuration**
- > The control opens the **Export LDAP configuration file window.**
- ▶ Enter the name for the server configuration file into the name field
- ▶ Save the file to the desired folder
- > The control exports the server configuration file.

Setting up LDAP on remote computer

To set up **LDAP on remote computer**:

- ▶ Opening the **User management** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **LDAP on remote computer**
- ▶ Select **Import server configuration**
- > The control opens the **Import LDAP configuration file window.**
- ▶ Select the existing configuration file
- ▶ Select **FILE**
- ▶ Select **APPLY**
- > The control imports the configuration file.

41.4.4 Connection to Windows domain

Application

With the **Connection to Windows domain** function you can connect the data of a domain controller with the control's user administration.

Related topics

- Configuring an LDAP database on a control

  Further information: "Local LDAP database", Page 2170
- Using an LDAP database on multiple controls

  Further information: "LDAP database on a remote computer", Page 2171

Requirements

- User administration is active

  Further information: "Activating user administration", Page 2165
- **useradmin** user is logged on

  Further information: "Users", Page 2160
- Windows domain controller present in the network
- You have access to the password of the domain controller
- You have access to the user interface of the domain controller, perhaps supported by an IT administrator
- Domain controller accessible in the network

Description of function

Use the **Configuration** function to configure the connection:

- Use the **Map SIDs to Unix UIDs** check box to select whether Windows SIDs are automatically mapped to Unix UIDs
- Use the **Use LDAPs** check box to select LDAP or secure LDAPs. For LDAPs, define whether or not the secure connection verifies a certificate
- Define a special group of Windows users to whom you want to restrict the connection to this control
- Modify the organizational unit in which the HEROS role names are stored
- Change the prefix in order to manage users from different workshops, for example. Each prefix given to a HEROS role name can be changed (e.g., HEROS hall 1 and HEROS hall 2)
- Modify the separator within the HEROS role names

Groups of the domain

If not all of the required roles have been created in the domain as groups, the control issues a warning.

If the control issues a warning, proceed in one of the two following ways:

- Use the **Add role definition** function to enter a role directly in the domain
- Use the **Export** function to export the roles to an ***.ldif** file

There are the following ways to create groups corresponding to the different roles:

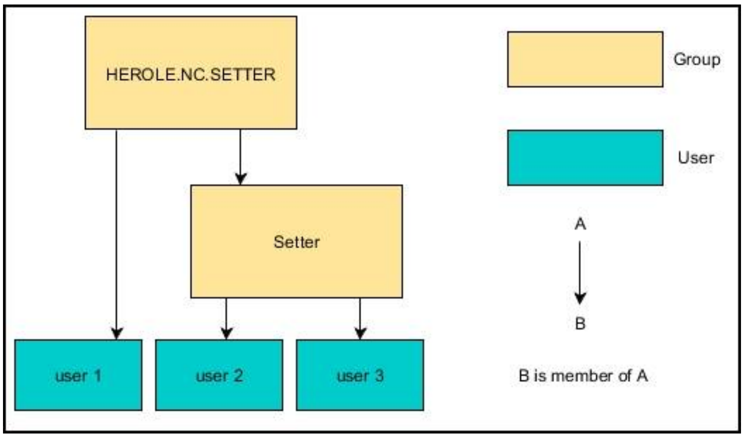
- Automatically when entering the Windows domain by specifying a user with administrator rights
- By importing an import file in .ldif format to the Windows server

The Windows administrator must add the users manually to the roles (security groups) on the domain controller.

Two suggestions describing how the groups can be structured by the Windows administrator are given by below.

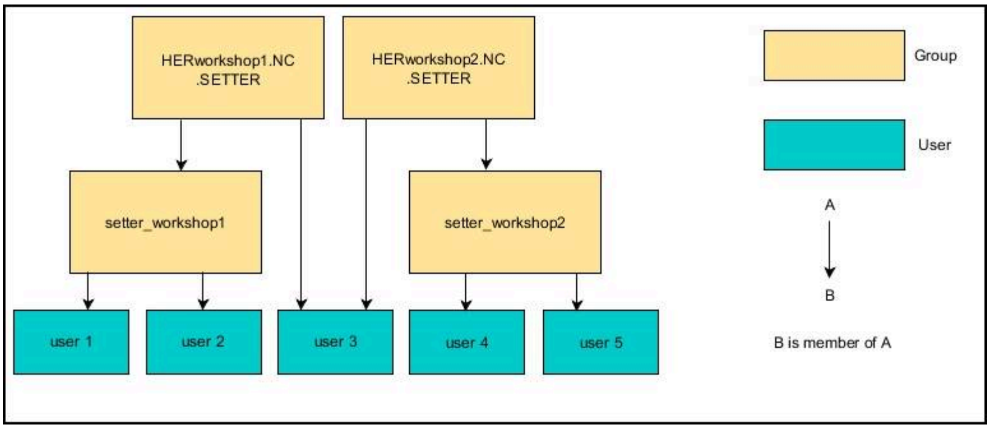
Example 1

The user is a direct or indirect member of the respective group:



Example 2

Users from various sectors (workshops) are members of groups with different prefixes:



Setting up the Connection to Windows domain function

To set up a **Connection to Windows domain**:

- ▶ Opening the **User management** window
- ▶ Select **Connection to Windows domain**
- ▶ Select **Find domain**
- > The control selects a domain.
- ▶ Select **APPLY**
- > The control opens the **Connect to domain** window.

> **i** With the **Organizational unit for computer account:** function, you can specify in which of the already existing organizational units you want to create the access, e.g.
> - ou=controls
> - cn=computers
>
> The values you enter must match the conditions of the domain. The terms are not exchangeable.

- ▶ Enter the user name of the domain controller
- ▶ Enter the password of the domain controller
- ▶ Confirm your input
- > The control connects to the Windows domain found.
- > The control checks whether all of the required roles have been created in the domain as groups.
- ▶ Add groups, if necessary

  Further information: "Groups of the domain", Page 2173

41.5 Autologin in user administration

Application

If the **Autologin** function is enabled, during startup the control automatically logs on a selected user without the need to enter a password.

As opposed to the **legacy mode**, this enables you to restrict a user's rights without entering a password.

Related topics

- User login
  Further information: "Logging on with user administration", Page 2175
- Configure user administration
  Further information: "Configuring user administration", Page 2164

Requirements

- User administration has been configured
- The user for **Autologin** has been defined

Description of function

With the **Enable autologin** check box in the **User management** window you can define a user for autologin.

Further information: "User management window", Page 2168

The control then automatically logs this user on and displays the user interface according to the defined rights.

For further authorizations, the control still requires an authentication to be entered.

Further information: "Window for requesting additional rights", Page 2177

41.6 Logging on with user administration

Application

The control displays a dialog window for user logon. Within the dialog the user can log on with a password or a smartcard.

Related topics

- Automatic user logon
  Further information: "Autologin in user administration", Page 2175

Requirements

- User administration has been configured
- For logon with smartcards:
  - Euchner EKS card reader
  - Smartcard assigned to a user
    Further information: "Assigning a smartcard to a user", Page 2177

Description of function

The control displays the Login dialog in the following cases:

- After the **User logout** function has been executed
- After the **Switch user** function has been executed
- After the screen has been locked by the **screensaver**
- Immediately after control startup if user administration is active and **Autologin** is not enabled

Further information: "HEROS menu", Page 2184

The logon dialog gives you the following options:

- Users who logged in at least once
- **Other** user

Logon with smartcards

You can save a user's logon data on a smartcard and then log the user on with a card reader, without needing to enter a password. You can define whether a PIN is necessary for logon.

The card reader is attached over a USB port. You assign the smartcard to a reader as a token.

Further information: "Assigning a smartcard to a user", Page 2177

The smartcard also has additional memory space, where the machine manufacturer can store his own user-specific data.

41.6.1 Logging on a user with password

To logon a user the first time:

- ▶ Select **Other** in the login dialog
- > The control enlarges the user icon you selected.
- ▶ Enter the user name
- ▶ Enter the user's password

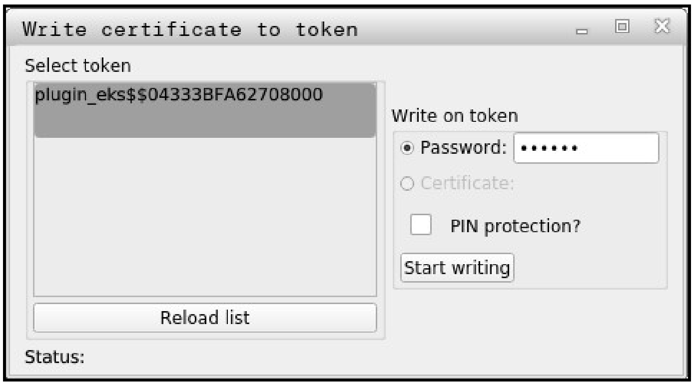
> The control shows in the Login dialog whether CAPS LOCK is active.

- > The control opens a window with the message **Password expired. Change the password now.**
- ▶ Enter the current password
- ▶ Enter a new password
- ▶ Repeat the new password
- > The control uses the new user to log you in.
- > The control displays this user in the dialog during the next logon procedure.

41.6.2 Assigning a smartcard to a user

To assign a smartcard to a user:

- ▶ Insert a blank smartcard in the card reader
- ▶ Logon the desired smartcard user in user administration
- ▶ Select the **Settings** application
- ▶ Select **Operating system**
- ▶ Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- ▶ Select **Create token**
- > The control opens the **Write certificate to token** window.
- > The control displays the smartcard in the **Select token** area.
- ▶ Select the smartcard as the token to be written
- ▶ Activate the **PIN protection?** check box, if desired
- ▶ Enter user password (and PIN, if desired)
- ▶ Select **Start writing**
- > The control saves the user's logon data on the smartcard.



Notes

- You must restart the control in order for it to detect a card reader.
- You can overwrite smartcards that already contain information.
- If you change a user's password, you must reassign the smartcard.

41.7 Window for requesting additional rights

Application

If you do not have the rights required for a specific item, the control opens the window for requesting additional rights.

In this window, you can temporarily obtain more rights by adding another user's rights.

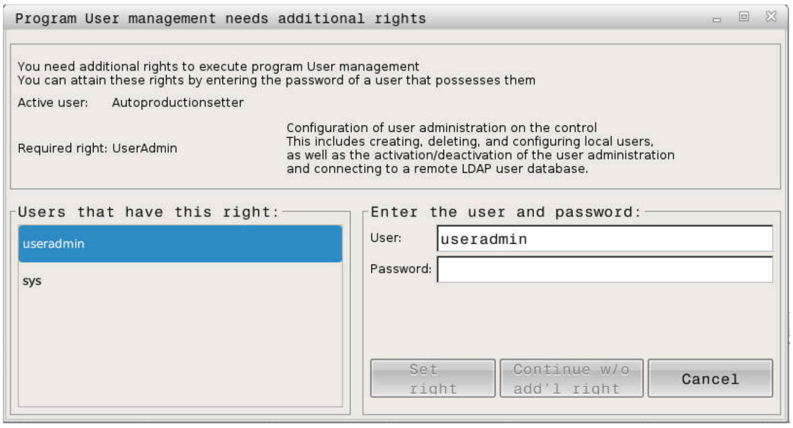
Related topics

- Temporarily granting additional rights in the **Active user** window
  Further information: "Active user window", Page 2168

Description of function

In the **Users that have this right:** field, the control lists all existing users that have the right to use this function.

You must enter the password in order to enable user rights.



Window for requesting additional rights

To attain the rights of users that are not shown, enter their user data. The control will then recognize those users that are contained in the user database.

Notes

- If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.
- You can't use this window to change user administration settings. The user with the HEROS.Admin role must be logged on in order to do so.

41.8 SSH-secured DNC connection

Application

If user administration is active, external applications also need to authenticate a user so that the suitable rights can be assigned.

For DNC connections using the RPC or LSV2 protocol, the connection is routed through an SSH tunnel. This method assigns the remote user to a user set up on the control, granting the remote user this user's rights.

Related topics

- Forbidding non-secure connections

  Further information: "Firewall", Page 2143
- Roles for remote logon

  Further information: "Roles", Page 2162

Requirements

- TCP/IP network
- The remote computer acts as SSH client
- The control acts as SSH server
- Key pair consisting of
  - Private key
  - Public key

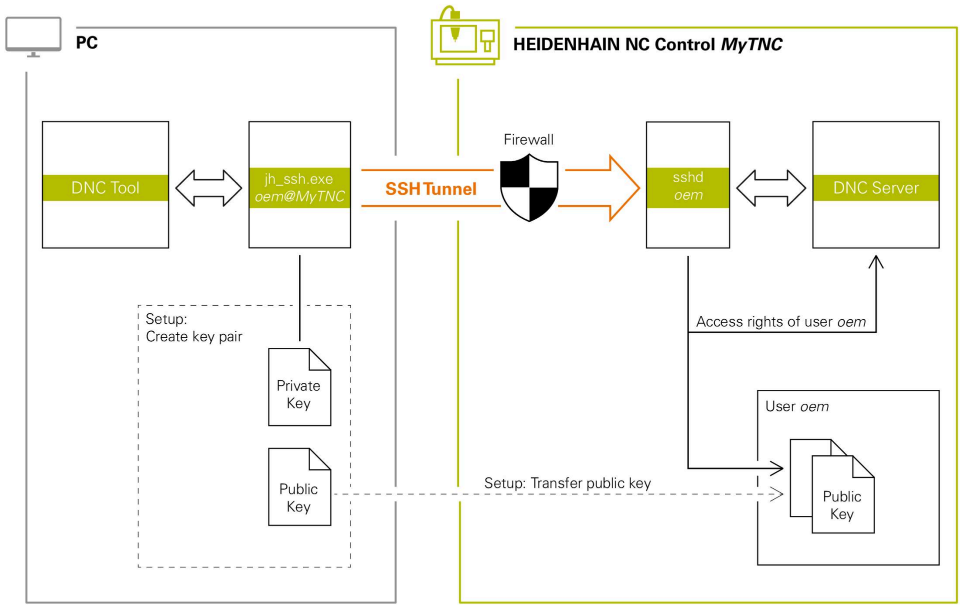
Description of function

Concept of transmission through an SSH tunnel

An SSH connection is always set up between an SSH client and an SSH server.

A key pair is used to protect the connection. This key pair is generated on the client. The key pair consists of a private key and a public key. The private key remains with the client. During setup, the public key is transferred to the server and assigned to a certain user.

The client tries to connect to the server using the pre-defined user name. The server can use the public key to verify that the requester of the connection holds the associated private key. If yes, the server accepts the SSH connection and assigns it to the user that has been used for the login. Communication can then be "tunneled" through this SSH connection.



Use in external applications

The PC tools available from HEIDENHAIN, such as TNCremo with version **v3.3** or higher, provide all functions for setting up, establishing, and managing secure connections through an SSH tunnel.

When the connection is set up, the required key pair is generated in TNCremo and the public key is transferred to the control.

This also applies to applications that are using the HEIDENHAIN DNC component from RemoTools SDK for communication. There is no need to adapt existing customer applications.

> In order to expand the connection configuration using the associated **CreateConnections** tool, you need to update to **HEIDENHAIN DNC v1.7.1**. A modification of the application source code is not required.

41.8.1 Setting up SSH-secured DNC connections

To set up an SSH-secured DNC connection for the logged-on user:

- Select the **Settings** application
- Select **Network/Remote Access**
- Select **DNC**
- Activate the **Setup permitted** toggle switch
- Use **TNCremo** to set up the secure connection (TCP secure).

> For details, refer to the integrated help system of TNCremo.

\> TNCremo transmits the public key to the control.

> In order to ensure maximum security, deactivate the **Allow password authentication** function after the public key has been stored.

- Deactivate the **Setup permitted** toggle switch

41.8.2 Removing a secure connection

If you delete a private key from the control, that user no longer has the possibility of a secure connection.

To delete a key:

- Select the **Settings** application
- Select **Operating system**
- Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- Select **Certificate and keys**
- Select the key to be deleted
- Select **Delete SSH key**
- > The control deletes the selected key.

Notes

- The the encryption used with the SSH tunnel protects the communication from attackers.
- For OPC UA connections, a stored user certificate is used for authentication.

  Further information: "OPC UA NC Server (options 56 to 61)", Page 2123
- 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 identification.

  The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).
- Once the connection configurations have been set up, they can be shared among all HEIDENHAIN PC tools for establishing a connection.
- You can also transmit a public key to the control using a USB device or a network drive.
- 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 needing to transmit them to the control.

42

HEROS Operating System

42.1 Fundamentals

HEROS is the fundamental basis for all NC controls from HEIDENHAIN. The HEROS operating system is based on Linux, and was adapted for the purposes of NC controls.

The TNC7 features the version HEROS 5.

42.2 HEROS menu

Application

In the HEROS menu the control shows information about the operating system. You can change settings or use HEROS functions.

By default you open the HEROS menu through the taskbar at the bottom edge of the screen

Related topics

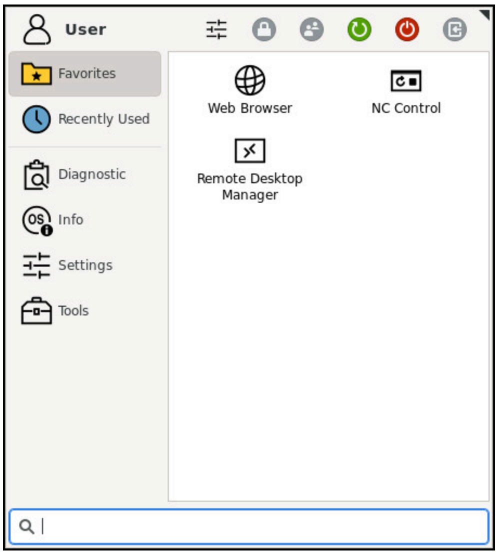
- Opening HEROS functions through the **Settings** application

  Further information: "Settings Application", Page 2099

Description of function

You open the HEROS menu with the green DIADUR icon in the taskbar or with the **DIADUR** key.

Further information: "Taskbar", Page 2188



Standard view of the HEROS menu

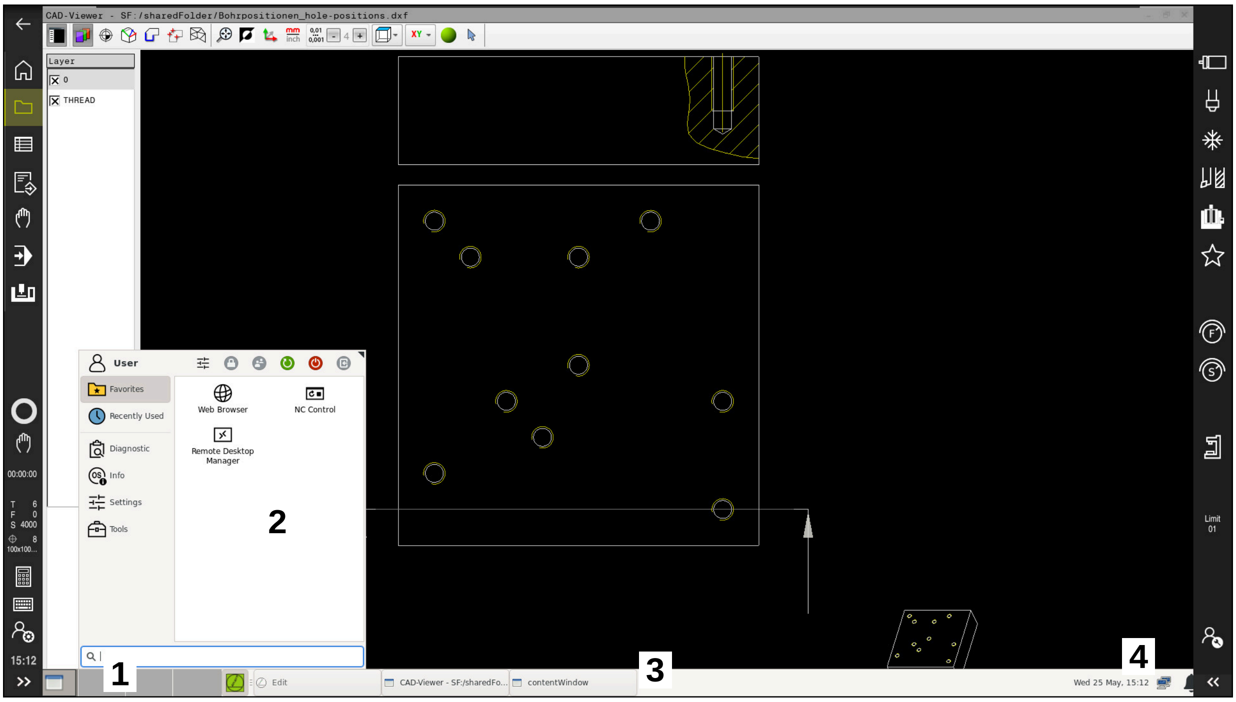
The HEROS menu contains the following functions:

Area	Function
Header	■ User name **Further information:** "Active user window", Page 2168 ■ User-specific settings ■ Lock display Only if user administration is active ■ Switch users Only if user administration is active ■ Restart ■ Shut down ■ Log out Only if user administration is active **Further information:** "User administration", Page 2159
Navigation	■ Favorites ■ Recently used
Diagnostic	■ **GSmartControl**: Available only to authorized specialists ■ **HeLogging**: Define settings for internal diagnostic files ■ **HeMenu**: Available only to authorized specialists ■ **perf2**: Check processor load and process load ■ **Portscan**: Test active connections **Further information:** "Portscan", Page 2146 ■ **Portscan OEM**: Available only to authorized specialists ■ **RemoteService**: Start and stop remote maintenance **Further information:** "Remote servicing", Page 2147 ■ **Terminal**: Enter and execute console commands ■ **TNCdiag**: Evaluates status and diagnostic information of HEIDENHAIN components with a focus on the drives and presents it graphically **Further information:** "TNCdiag", Page 2152 ■ **TNCscope** Software for data recording

Area	Function
Settings	■ **Screensaver**: Screensaver ■ **Current User** **Further information:** "Active user window", Page 2168 ■ **Date/Time** **Further information:** "Adjust system time window", Page 2110 ■ **Firewall** **Further information:** "Firewall", Page 2143 ■ **HePacketManager**: Available only to authorized specialists ■ **HePacketManager Custom**: Available only to authorized specialists ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 2111 ■ **Network** **Further information:** "Ethernet interface", Page 2116 ■ **OEM Function Users** **Further information:** "User administration", Page 2159 ■ **OPC UA NC Server Connection Assistant** **Further information:** "OPC UA connection wizard function (options 56 to 61)", Page 2126 ■ **OPC UA NC Server License** **Further information:** "OPC UA license settings function (options 56 to 61)", Page 2127 ■ **PKI Admin**: Manage certificates for the control (e.g., for **OPC UA NC Server**) "OPC UA NC Server (options 56 to 61)" ■ **Printer** **Further information:** "Printers", Page 2130 ■ **SELinux** **Further information:** "SELinux security software", Page 2112 ■ **Shares** **Further information:** "Network drives on the control", Page 2113 ■ **UserAdmin** **Further information:** "User management window", Page 2168 ■ **VNC** **Further information:** "VNC menu item", Page 2133 ■ **WindowManagerConfig**: Settings for the Window Manager **Further information:** "Window Manager", Page 2189
Info	■ **About HeROS**: Open information about the operating system of the control ■ **About Xfce**: Open information on the Window manager

Area	Function
Tools	■ **Switch-off**: Shut-down or restart ■ **Screenshot**: Create screenshots ■ **File Manager**: Available only to authorized specialists ■ **Document Viewer**: Display and print files (e.g., PDF files) ■ **Geeqie**: Open, manage, and print graphics ■ **Gnumeric**: Open, edit, and print tables ■ **IDS Camera Manager**: Manage cameras connected to the control ■ **keypad horizontal**: Open virtual keyboard ■ **keypad vertical**: Open virtual keyboard ■ **Leafpad**: Open and edit text files ■ **NC Control**: Start or stop the NC software independently of the operating system ■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 2148 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 2148 ■ **QupZilla**: Alternative web browser for touch operation ■ **Real VNC Viewer**: Define the settings for external software accessing the control (e.g., for maintenance purposes) ■ **Remote Desktop Manager** **Further information:** "Remote Desktop Manager window (option 133)", Page 2137 ■ **Ristretto**: Open graphics ■ **TNCguide**: Open help files in CHM format ■ **TouchKeyboard**: Open keyboard for touch operation ■ **Web Browser**: Start the web browser ■ **Xarchiver**: Extract or compress directories
Searching	Full-text search of individual functions

Taskbar



CAD-Viewer opened in the third desktop with taskbar shown and active HEROS menu

The taskbar consists of the following areas:

1 Workspaces
2 HEROS menu
   Further information: "Description of function", Page 2184
3 Opened applications, e.g.:
   - Control interface
   - **CAD-Viewer**
   - Window of HEROS functions

   You can move the opened applications into any other workspaces.
4 Widgets
   - Calendar
   - Status of the firewall

     Further information: "Firewall", Page 2143
   - Network status

     Further information: "Ethernet interface", Page 2116
   - Notifications
   - Shut down or restart the operating system

Window Manager

With the Window Manager you manage functions of the HEROS operating system as well as windows opened in the third desktop, such as the **CAD-Viewer**.

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

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

If a window is opened in the third desktop, the control displays the **Window Manager** icon in the information bar. You can switch between the open applications by selecting the icon.

You can minimize the control's user interface by pulling down from the information bar. The TNC bar and the OEM bar remain visible.

Further information: "Areas of the control's user interface", Page 109

Notes

- If a window is opened in the third desktop, the control displays an icon in the information bar.

  Further information: "Areas of the control's user interface", Page 109
- The machine manufacturer determines the scope of function and behavior of the window manager.
- The control shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

42.3 Serial data transfer

Application

The TNC7 automatically uses the LSV2 transmission protocol for serial data transfer. All parameters of the LSV2 protocol are invariably fixed except for the baud rate in the machine parameter **baudRateLsv2** (no. 106606).

Description of function

The machine parameter **RS232** (no. 106700) allows you to define another transmission type (interface). The settings described below are effective only for the respective newly defined interface.

Further information: "Machine parameters", Page 2152

In the machine parameters that then appear you can define the following settings:

Machine parameters	Setting
baudRate (no. 106701)	Data transfer rate (baud rate) Input: **BAUD_110**, **BAUD_150**, **BAUD_300 BAUD_600**, **BAUD_1200**, **BAUD_2400**, **BAUD_4800**, **BAUD_9600**, **BAUD_19200**, **BAUD_38400**, **BAUD_57600**, **BAUD_115200**
protocol (no. 106702)	Communications protocol ■ **STANDARD**: Standard data transmission, line-by-line ■ **BLOCKWISE**: Packet-based data transfer ■ **RAW_DATA**: Transmission without protocol (purely character-by-character) Input: **STANDARD**, **BLOCKWISE**, **RAW_DATA**
dataBits (no. 106703)	Data bits in each transferred character Input: **7 Bit**, **8 Bit**
parity (no. 106704)	Parity bit used to check for transmission errors ■ **NONE**: No parity, no error detection ■ **EVEN**: Even parity, error if the number of bits set is odd ■ **ODD**: Odd parity, error if the number of bits set is even Input: **NONE**, **EVEN**, **ODD**
stopBits (no. 106705)	The start bit and one or two stop bits enable the receiver to synchronize each transmitted character during serial data transmission. Input: **1 Stop-Bit**, **2 Stop-Bits**
flowControl (no. 106706)	By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking. ■ **NONE**: No data-flow check ■ **RTS_CTS**: Hardware handshaking, transmission stop is active through RTS ■ **XON_XOFF**: Software handshaking, transmission stop is active through DC3 Input: **NONE**, **RTS_CTS**, **XON_XOFF**
fileSystem (no. 106707)	File system for the serial interface ■ **EXT**: Minimum file system for printers or non-HEIDENHAIN transmission software ■ **FE1**: Communication with TNCserver or an external floppy disk unit If you require no special file system, this machine parameter is not needed. Input: **EXT**, **FE1**
bccAvoidCtrlChar (no. 106708)	The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection. ■ **TRUE**: The BCC does not correspond to any control character ■ **FALSE**: Function not active Input: **TRUE**, **FALSE**

Machine parameters	Setting
rtsLow (no. 106709)	This optional parameter determines the level of the RTS line in the idle state. ■ **TRUE**: Level is **LOW** in idle state ■ **FALSE**: Level is **HIGH** in idle state Input: **TRUE**, **FALSE**
noEotAfterEtx (no. 106710)	This optional parameter sets whether an EOT character (End of Transmission) is to be transmitted after receiving an ETX character (End of Text). ■ **TRUE**: The EOT character is not sent ■ **FALSE**: The EOT character is sent Input: **TRUE**, **FALSE**

Example

In order to use the TNCserver PC software for data transfer, define the following settings in the machine parameter **RS232** (no. 106700):

Parameters	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Type of handshake	RTS_CTS
File system for file operations	FE1

TNCserver is part of the TNCremo software for PCs.

Further information: "PC software for data transfer", Page 2191

42.4 PC software for data transfer

Application

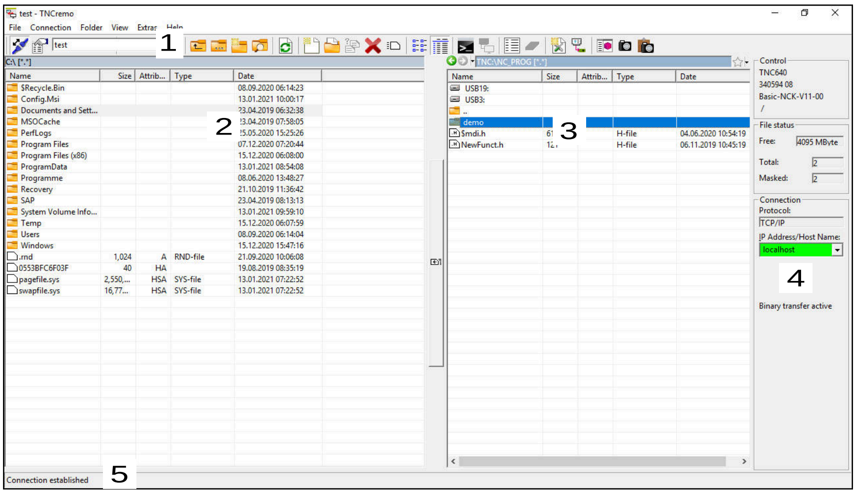
HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Requirements

- PC operating system:
  - Windows 7
  - Windows 8
  - Windows 10
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- An available serial interface or connection to the control's network

Description of function

The TNCremo data transfer software provides the following areas:



1 Toolbar

This area provides the most important TNCremo functions.

2 File list of PC

In this area, TNCremo displays all of the folders and files of the connected drive (e.g., hard disk of a Windows PC or a USB flash drive).

3 File list of control

In this area, TNCremo displays all of the folders and files of the connected drive of the control.

4 Status display

In the status display, TNCremo shows information about the current connection.

5 Connection status

The connection status indicates whether a connection is currently active.

> For more information, refer to the integrated help system of TNCremo.
> You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

Notes

- 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 identification. The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).

  The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).
- You can download the current version of the TNCremo software from the **HEIDENHAIN website**.

42.5 Data backup

Application

If you create or modify files on the control, then you should back up these files periodically.

Related topics

- File management
  Further information: "File management", Page 1144

Description of function

With the functions **NC/PLC Backup** and **NC/PLC Restore** you can create back-up files for specific directories or even an entire drive, and restore them as needed. You should store these backup files on an external storage medium.

Further information: "Backup and restore", Page 2148

You have the following options for transferring files from the control:

- TNCremo
  With TNCremo you can transfer files from the control to a computer.
  Further information: "PC software for data transfer", Page 2191
- External drive
  You can transfer files from the control directly to an external drive.
  Further information: "Network drives on the control", Page 2113
- External data carriers
  You can back-up files to external data carriers or use external data carriers to transfer the files.
  Further information: "USB devices", Page 1157

Notes

- You should back-up all machine-specific data, such as the PLC program or machine parameters. Consult your machine manufacturer about this.
- You must transmit files with the extensions PDF, XLS, ZIP, BMP, GIF, JPG and PNG in binary format from the PC to the control's hard disk.
- Backing up all files of the internal memory can take several hours. If required, perform the backup during a time when you don't need the machine.
- Periodically delete files that are no longer required. This ensures that the control has enough memory available for system files, such as the tool table.
- HEIDENHAIN recommends having the hard disk inspected after three to five years. After this time, and depending on the operating conditions (e.g., vibration loads), you must expect increased failure rates.

42.6 Opening files with additional software

Application

The control provides several additional software programs for opening and editing standard file types:

Related topics

- File types
  Further information: "File types", Page 1149

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XLSX (XLS) CSV	Gnumeric
INI A TXT	Leafpad
HTM/HTML	Web browser For networks and the Internet, the machine manufacturer or network administrator must guarantee that the control is protected against viruses and malware (e.g., by a firewall).
ZIP	Xarchiver
BMP GIF JPG/JPEG PNG	Ristretto or Geeqie Ristretto can only open graphics files. Geeqie can also edit and print graphics.
OGG	Parole With Parole you can open the file types OGA, OGG, OGV and OGX. The Fuendo Codec Pack (available for payment) is needed only for other formats, such as MP4 files.

If you double-tap or double-click a file in the file manager, the control automatically starts the file with the correct tool. If more than one tool is possible for a file, the control displays a selection window.

The control opens the tools in the third desktop.

42.6.1 Opening tools

To open a tool:

- ▶ Select the HEIDENHAIN icon in the taskbar
- > The control opens the HEROS menu.
- ▶ Select **Tools**
- ▶ Select the tool (e.g. **Leafpad**)
- > The control opens the tool in its own workspace.

Notes

- You can also open several tools from the **Desktop menu** workspace.
- Use the **ALT+TAB** key combination to switch between open workspaces.
- More information on how to use the various tools is provided within the respective tool under Help.
- After starting, the **web browser** checks at regular intervals whether updates are available.

  If you want to update the **web browser**, then you must deactivate the SELinux security software during this time and establish a connection to the Internet. Reactivate SELinux after the update!

  Further information: "SELinux security software", Page 2112

42.7 Network configuration with Advanced Network Configuration

Application

Using the **Advanced Network Configuration**, you can add, edit or remove profiles for the network connection.

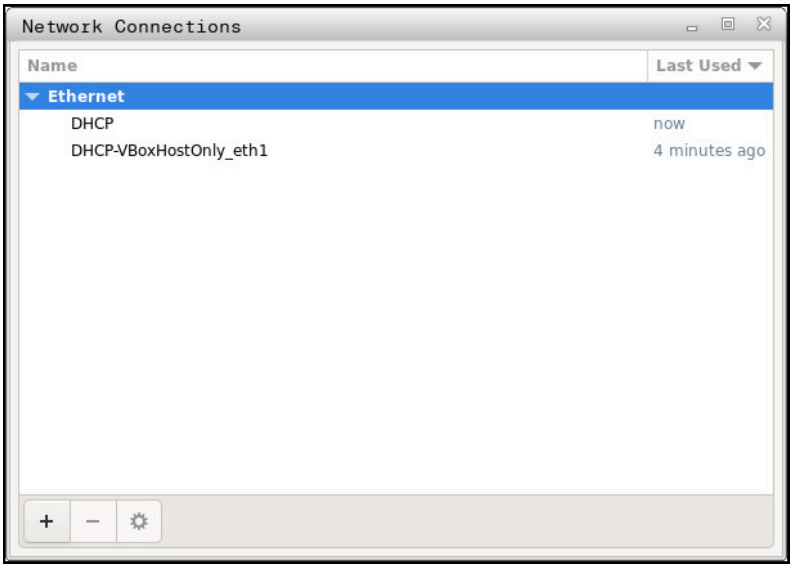
Related topics

- Network settings

  Further information: "Editing network connection window", Page 2197

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network Connections** window.



Network Connections window

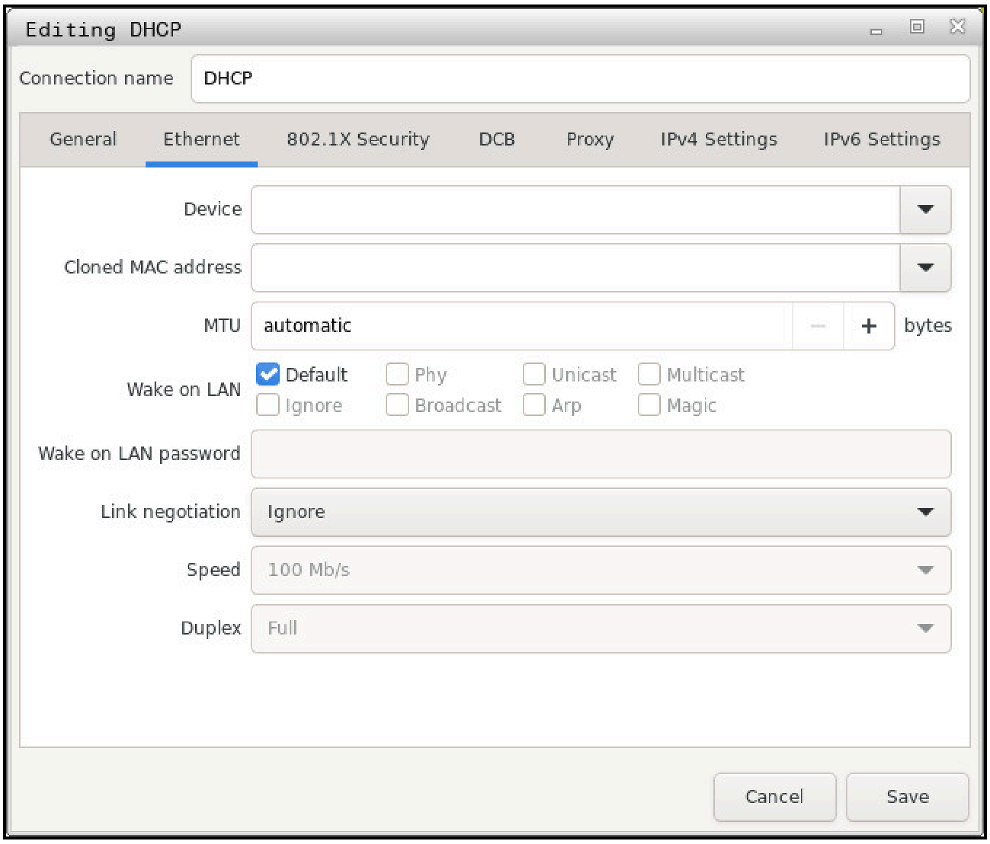
Symbols in the Network connections window

The following symbols are shown in the **Network connections** window:

Symbol	Function
+	Add network connection
−	Remove network connection
⚙	Edit network connection The control opens the **Editing network connection** window. **Further information:** "Editing network connection window", Page 2197

42.7.1 Editing network connection window

In the **Editing network connection** window, the control shows the connection name of the network connection in the upper area. You can change the name.



Editing network connection window

General tab

The **General** tab contains the following settings:

Setting	Meaning
Connect _automatically	If you are using several profiles, you can define an order of priority for the connection here. The control connects the network with the highest priority first. Input: **-999...999**
All _users may connect to this network	Here you can enable the selected network for all users.
Automatically connect to _VPN when using this connection	Currently no function
Bonded _connections:	Currently no function

Ethernet tab

The **Ethernet** tab contains the following settings:

Setting	Meaning
_Service:	Here you can select the Ethernet interface. If you do not select an Ethernet interface, this profile can be used for any Ethernet interface. Selection by means of a selection window
_Cloned MAC address:	Currently no function
_MTU:	Here you can define the maximum package size in bytes. Input: **Automatic**, **1...10000**
_Private key password:	Currently no function
Wake-on-LAN password	Currently no function
Lin_k negotiation	Here you have to configure the settings for the Ethernet connection: ■ **Ignore** Retain the configurations already existing on the device. ■ **Automatic** The speed and duplex settings are configured automatically for the connection. ■ **Manual** Configure the speed and duplex settings for the connection manually. Selection by means of a selection window
Speed	Here you have to select the speed settings: ■ **10 Mb/s** ■ **100 Mb/s** ■ **1 Gb/s** ■ **10 Gb/s** Only if **Lin_k negotiation Manual** is selected Selection by means of a selection window
Full duple_x	Here you have to select the duplex setting: ■ **Half** ■ **Full** Only if **Lin_k negotiation Manual** is selected Selection by means of a selection window

802.1X Security tab

Currently no function

DCB tab

Currently no function

Proxy tab

Currently no function

IPv4 Settings tab

The **IPv4 Settings** tab contains the following settings:

Setting	Meaning
_Method:	Here you have to select a network connection method: ■ **Automatic (DHCP)** If the network uses a DHCP server for IP address assignment ■ **Automatic (DHCP) addresses only** If the network uses a DHCP server for IP address assignment, but you are assigning the DNS server manually ■ **Manual** Assign the IP address manually ■ **Link-Local Only** Currently no function ■ **Shared to other computers** Currently no function ■ **Disabled** Deactivate IPv4 for this connection
Automatic, addresses only	Here you can add static IP addresses that will be set up in addition to the IP addresses that are assigned automatically. Only with **_Method: Manual**
Additional DNS ser_vers:	Here you can add the IP addresses of DNS servers that are used to resolve computer names. Separate multiple IP addresses by commas. Only with **_Method: Manual** and **Automatic (DHCP) addresses only**
Additional s_earch domains:	Here you can add domains used by computer names. Separate multiple domains by commas. Only with **_Method: Manual**
D_HCP client ID:	Currently no function
Require IPv_4 addressing for this connection to complete	Currently no function

IPv6 Settings tab

Currently no function

43

Overviews

43.1 Pin layout and cables for data interfaces

43.1.1 V.24/RS-232-C interface for HEIDENHAIN devices

The interface complies with the requirements of EN 50178 for Secure separation from the power grid.

Control		25-pin: VB 274545-xx			9-pin: VB 366964-xx		
Male	**Assignment**	**Male**	**Color**	**Female**	**Female**	**Color**	**Female**
1	Do not assign	1	White/Brown	1	1	Red	1
2	RXD	3	Yellow	2	2	Yellow	3
3	TXD	2	Green	3	3	White	2
4	DTR	20	Brown	8	4	Brown	6
5	Signal GND	7	Red	7	5	Black	5
6	DSR	6		6	6	Violet	4
7	RTS	4	Gray	5	7	Gray	8
8	CTR	5	Pink	4	8	White/Green	7
9	Do not assign	8	Violet	20	9	Green	9
Housing	External shield	Housing	External shield	Housing	Housing	External shield	Housing

43.1.2 Ethernet interface RJ45 socket

Maximum cable length:

- 100 m unshielded
- 400 m shielded

Pin	Signal
1	TX+
2	TX–
3	RX+
4	Vacant
5	Vacant
6	RX–
7	Vacant
8	Vacant

43.2 Machine parameters

The following list shows the machine parameters that you can edit with the code number 123.

Related topics

- Editing machine parameters with the **MPs for setters** application
  Further information: "Machine parameters", Page 2152

43.2.1 List of user parameters

Refer to your machine manual.

- The machine manufacturer can make additional machine-specific parameters available as user parameters, so that you can configure the functions that are available.
- The machine manufacturer can adapt the structure and contents of the user parameters. The display on your machine may be different.

Depiction in the configuration editor	MP number	Page
DisplaySettings		-
CfgDisplayData Settings for screen displays	100800	2214
axisDisplay Display sequence and display rules for axes	100810	2214
x		-
axisKey Key name of the axis	100810.[Index].01501	2214
name Axis designation	100810.[Index].01502	2214
rule Display rule for the axis	100810.[Index].01503	2214
axisDisplayRef Sequence and rules for display axes before crossing the reference marks	100811	2215
x		-
axisKey Key name of the axis	100811.[Index].01501	2215
name Axis designation	100811.[Index].01502	2215
rule Display rule for the axis	100811.[Index].01503	2216
positionWinDisplay Type of position display in the position window	100803	2216
statusWinDisplay Type of position display in the Status workspace	100804	2217
decimalCharacter Definition of the decimal separator for the position display	100805	2217
axisFeedDisplay Display of the feed rate in the applications of the Manual operating mode	100806	2217
spindleDisplay Display of spindle position in the position display	100807	2218

Depiction in the configuration editor	MP number	Page
hidePresetTable Disable the PRESET MANAGEMENT soft key	100808	2218
displayFont Font size for program display in the operating modes Program Run Full Sequence, Program Run Single Block, and Positioning with Manual Data Input.	100812	2218
iconPrioList Sequence of icons in the display	100813	2218
compatibilityBits Settings for display behavior	100815	2219
axesGridDisplay Axes as list or group in the position display.	100806	2219
CfgPosDisplayPace Display step for the individual axes	101000	2219
xx		-
displayPace Display step for position display in [mm] or [°]	101001	2219
displayPaceInch Display step for position display in [inch]	101002	2220
CfgUnitOfMeasure Definition of unit of measure in effect for display	101100	2220
unitOfMeasure Unit of measure for display and user interface	101101	2220
CfgProgramMode Format of the NC programs and cycle display	101200	2220
programInputMode MDI: Program entry in HEIDENHAIN Klartext format or ISO format	101201	2221
CfgDisplayLanguage Definition of the NC and PLC conversational language	101300	2221
ncLanguage NC conversational language	101301	2221
applyCfgLanguage Load the language of the NC control	101305	2222
plcDialogLanguage PLC conversational language	101302	2222
plcErrorLanguage PLC error message language	101303	2223
helpLanguage Language for online help	101304	2223

Depiction in the configuration editor	MP number	Page
CfgStartupData Behavior during control startup	101500	2224
powerInterruptMsg Acknowledge the Power interrupted message	101501	2224
opMode Operating mode that is switched to when the control has fully booted	101503	2224
subOpMode Submode to be activated for the operating mode entered in 'opMode'	101504	2224
CfgClockView Display mode for time of day	120600	2225
displayMode Display mode for time of day on the screen	120601	2225
timeFormat Time format of digital clock	120602	2225
CfgInfoLine Link row on/off	120700	2225
infoLineEnabled Enable/disable info line	120701	2225
CfgGraphics Settings for 3D simulation graphics	124200	2226
modelType Model type of the 3D simulation graphics	124201	2226
modelQuality Model quality of the 3D simulation graphics	124202	2226
clearPathAtBlk Reset tool paths for new BLK FORM	124203	2226
extendedDiagnosis Write graphics journal files after restart	124204	2227
CfgPositionDisplay Settings for the digital readout	124500	2227
progToolCallDL Position display with TOOL CALL DL	124501	2227
CfgTableEditor Table editor configuration	125300	2227
deleteLoadedTool Behavior when deleting tools from the pocket table	125301	2227
indexToolDelete Behavior when deleting a tool's index entries	125302	2228
showResetColumnT Show the RESET COLUMN T soft key	125303	2228

Depiction in the configuration editor	MP number	Page
CfgDisplayCoordSys Setting the coordinate systems for the display	127500	2228
transDatumCoordSys Coordinate system for the datum shift	127501	2228
CfgGlobalSettings GPS display settings	128700	2229
enableOffset Show offset in the GPS dialog	128702	2229
enableBasicRot Show an additive basic rotation in the GPS dialog	128703	2229
enableShiftWCS Show shift of W-CS in the GPS dialog	128704	2229
enableMirror Show mirroring in the GPS dialog	128712	2230
enableShiftMWCS Show shift of mW-CS in the GPS dialog	128711	2230
enableRotation Show rotation in the GPS dialog	128707	2230
enableFeed Show feed rate in the GPS dialog	128708	2230
enableHwMCS M-CS coordinate system is selectable	128709	2230
enableHwWCS W-CS coordinate system is selectable	128710	2231
enableHwMWCS mW-CS coordinate system is selectable	128711	2231
enableHwWPLCS WPL-CS coordinate system is selectable	128712	2231
enableHwAxisU U axis can be selected	128709	2231
enableHwAxisV V axis can be selected	128709	2232
enableHwAxisW W axis can be selected	128709	2232
CfgRemoteDesktop Settings for Remote Desktop connections	100800	2232
connections List of Remote Desktop connections to be displayed	133501	2232
autoConnect Start connection automatically	133505	2232

Depiction in the configuration editor	MP number	Page
title Name of the OEM operating mode	133502	2232
dialogRes Name of a text	133502.00501	2233
text Language-sensitive text	133502.00502	2233
icon Path/name for optional icon graphic file	133503	2233
locations List with positions where this Remote Desktop connection is displayed	133504	2233
x		-
opMode Operating mode	133504.[Index].133401	2233
subOpMode Optional submode for the operating mode specified in 'opMode'	133504.[Index].133402	2233
PalletSettings		-
CfgPalletBehaviour Behavior of the pallet control cycle	202100	2234
failedCheckReact Specify reaction to program check and tool check	202106	2234
failedCheckImpact Specify effect of program check or tool check	202107	2234
ProbeSettings		-
CfgTT Configuration of the tool calibration	122700	2235
TT140_x		-
spindleOrientMode M function for spindle orientation	122704	2235
probingRoutine Probing routine	122705	2235
probingDirRadial Probing direction for tool radius measurement	122706	2235
offsetToolAxis Distance from lower edge of tool to upper edge of stylus	122707	2236
rapidFeed Rapid traverse in probing cycle for TT tool touch probe	122708	2236

Depiction in the configuration editor	MP number	Page
probingFeed Probing feed rate for tool measurement with non-rotating tool	122709	2236
probingFeedCalc Calculation of the probing feed rate	122710	2236
spindleSpeedCalc Speed determination method	122711	2236
maxPeriphSpeedMeas Maximum permissible surface speed of the tool edge for radius measurement	122712	2237
maxSpeed Maximum permissible speed during tool measurement	122714	2237
measureTolerance1 Maximum permissible measuring error for tool measurement with rotating tool (first measurement error)	122715	2237
measureTolerance2 Maximum permissible measuring error for tool measurement with rotating tool (second measurement error)	122716	2237
stopOnCheck NC stop during tool check	122717	2237
stopOnMeasurement NC stop during tool measurement	122718	2238
adaptToolTable Change the tool table during tool check and tool measurement	122719	2238
CfgTTRoundStylus Configuration of a round stylus	114200	2238
TT140_x		-
centerPos Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum	114201	2238
safetyDistToolAx Safety clearance around the probe contact of the TT tool touch probe for pre-positioning in the tool-axis direction	114203	2239
safetyDistStylus Safety zone around the stylus for pre-positioning	114204	2239
CfgTTRectStylus Configuration of a rectangular stylus	114300	2239
TT140_x		-

Depiction in the configuration editor	MP number	Page
centerPos Coordinates of the stylus center	114313	2239
safetyDistToolAx Set-up clearance above the stylus for pre-positioning	114317	2239
safetyDistStylus Safety zone around the stylus for pre-positioning	114318	2239
ChannelSettings		-
CH_xx		-
CfgActivateKinem Active kinematics	204000	2241
kinemToActivate Kinematics to be activated / active kinematics	204001	2241
kinemAtStartup The kinematics to be activated during control start-up	204002	2241
CfgNcPgmBehaviour Specify the behavior of the NC program.	200800	2241
operatingTimeReset Reset the machining time when program starts.	200801	2241
plcSignalCycle PLC signal for the number of the pending machining cycle	200803	2242
CfgGeoTolerance Geometry tolerances	200900	2242
circleDeviation Permissible deviation of the radius	200901	2242
threadTolerance Permissible deviation in successive threads	200902	2242
moveBack Reserve for retraction movements	200903	2242
CfgGeoCycle Configuration of the fixed cycles	201000	2242
pocketOverlap Overlap factor for pocket milling	201001	2243
posAfterContPocket Traverse after machining the contour pocket	201007	2243

Depiction in the configuration editor	MP number	Page
displaySpindleErr Display the Spindle is not rotating error message if M3/M4 is not active	201002	2243
displayDepthErr Display the Check the depth sign error message	201003	2243
apprDepCylWall Behavior when moving to wall of slot in the cylinder surface	201004	2244
mStrobeOrient M function for spindle orientation in machining cycles	201005	2244
suppressPlungeErr Do not show 'Plunging type is not possible' error message	201006	2244
restoreCoolant Behavior of M7 and M8 with Cycles 202 and 204	201008	2244
facMinFeedTurnSMAX Automatic feed rate reduction after attaining SMAX	201009	2245
suppressResMatlWar Do not show "Residual material" warning	201010	2245
CfgStretchFilter Geometry filter for filtering out linear elements	201100	2245
filterType Type of stretch filter	201101	2246
tolerance Maximum distance of the filtered to the unfiltered contour	201102	2246
maxLength Maximum length of the distance resulting from filtering	201103	2246
CfgThreadSpindle Special spindle parameters for threads	113600	2246
sourceOverride Effective override potentiometer for feed rate during thread cutting	113603	2246
thrdWaitingTime Waiting time at reversal point in thread base	113601	2247
thrdPreSwitchTime Advanced switching time of spindle	113602	2247

Depiction in the configuration editor	MP number	Page
limitSpindleSpeed Limit of spindle speed with Cycles 17, 207 and 18	113604	2247
CfgEditorSettings Settings for the NC editor	105400	2248
createBackup Generate a backup file *.bak	105401	2248
deleteBack Behavior of the cursor after deletion of lines	105402	2248
lineBreak Line break on NC blocks with more than one line	105404	2248
stdTNChelp Activate help graphics when entering cycle data	105405	2248
warningAtDEL Confirmation request when deleting an NC block.	105407	2249
maxLineGeoSearch Line number up to which a test of the NC program is to be run.	105408	2249
blockIncrement ISO programming: Block number increment	105409	2249
useProgAxes Specify programmable axes	105410	2249
enableStraightCut Allow or lock paraxial positioning blocks	105411	2250
noParaxMode Hide FUNCTION PARAXCOMP/PARAXMODE	105413	2250
CfgPgmMgt Settings for the file management	122100	2250
dependentFiles Display of dependent files	122101	2250
CfgProgramCheck Settings for tool-usage files	129800	2251
autoCheckTimeOut Timeout for creation of tool-usage files	129803	2251
autoCheckPrg Create tool-usage file for NC program	129801	2251
autoCheckPal Create pallet-usage files	129802	2251
CfgUserPath Paths for the end user	102200	2252
ncDir List of drives and/or directories	102201	2252

Depiction in the configuration editor	MP number	Page
fn16DefaultPath Default output path for the FN16: F-PRINT function in the Program Run operating modes	102202	2252
fn16DefaultPathSim Default output path for the FN16: F-PRINT function in the Programming and Test Run operating modes	102203	2252
serialInterfaceRS232		-
CfgSerialPorts Data record belonging to the serial port	106600	2253
activeRs232 Enable the RS-232 interface in the program manager	106601	2253
baudRateLsv2 Data transfer rate for LSV2 communication in baud	106606	2253
CfgSerialInterface Definition of data records for the serial ports	106700	2253
RSxxx		-
baudRate Data transfer rate for communication in baud	106701	2254
protocol Communications protocol	106702	2254
dataBits Data bits in each transferred character	106703	2254
parity Type of parity checking	106704	2255
stopBits Number of stop bits	106705	2255
flowControl Type of data-flow checking	106706	2255
fileSystem File system for file operation via serial interface	106707	2255
bccAvoidCtrlChar Avoid control characters in the block check character (BCC)	106708	2256
rtsLow Idle state of the RTS line	106709	2256
noEotAfterEtx Behavior after reception of an ETX control character	106710	2256
Monitoring		-

Depiction in the configuration editor	MP number	Page
CfgMonUser Monitoring settings for the user	129400	2257
enforceReaction The configured error reactions are enforced	129401	2257
showWarning Display warnings of monitoring tasks	129402	2257
CfgMonMbSection CfgMonMbSection defines monitoring tasks for a certain section of an NC program	02400	2257
tasks List of monitoring tasks to be performed	133701	2257
CfgMachineInfo General information of the machine operator	131700	2258
machineNickname Custom name (nickname) of the machine	131701	2258
inventoryNumber Inventory number or ID	131702	2258
image Photo or image of the machine	131703	2258
location Machine location	131704	2258
department Department or division	131705	2258
responsibility Responsible for the machine	131706	2258
contactEmail Contact email address	131707	2259
contactPhoneNumber Contact phone number	131708	2259

43.2.2 Details about the user parameters

Explanations about the detailed view of user parameters:

- The indicated path corresponds to the machine parameter structure that you see after entering the machine manufacturer code number. With this information you can also find the desired machine parameter in the alternative structure. With the machine parameter numbers you can search for the machine parameters independently of the structure.
- The entry after iTNC shows the machine parameter number on the iTNC 530.

DisplaySettings

CfgDisplayData	100800
Settings for screen displays	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData
Data object:	

axisDisplay	100810
Display sequence and display rules for axes	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay
Input:	List (vacant or index 0 to 23) Specifies the sequence and the rules for the display of axes. The top-most entry corresponds to the top-most position. Up to 24 entries with the parameters ■ axisKey ■ name ■ rule

axisKey	100810.[Index].01501
Key name of the axis	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay ▶ [Index] ▶ axisKey
Input:	Select the key name of the axis for which this display setting is valid. The key names of the axes are taken from the configuration object **CfgAxis** and displayed as a selection menu.

name	100810.[Index].01502
Axis designation	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay ▶ [Index] ▶ name
Input:	max. 2 Characters Define the axis designation that, as an alternative to the key name from **CfgAxis** is used for the display. If the parameter is not set, then the TNC7 displays the key name.

rule	100810.[Index].01503
Display rule for the axis	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay ▶ [Index] ▶ rule
Input:	Defines the condition under which the axis is displayed. **ShowAlways**

Axis is always shown. The display location remains reserved even if no values for the axis can be displayed, for example if the axis is not contained in the current kinematic model.

IfKinem

Axis is shown only if it is used as an axis or a spindle in the active kinematic model.

IfKinemAxis

Axis only shown if used as axis in the active kinematics model.

IfNotKinemAxis

The axis is only shown if it is not used as an axis in the active kinematics model (e.g. as spindle).

Never

The axis is not shown.

axisDisplayRef 100811

Sequence and rules for display axes before crossing the reference marks

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplayRef
Input:	List (vacant or index 0 to 23) Specifies the sequence and the rules for the display of axes if the position display is set to REF values (also applies when traversing to the reference point). If this list is empty, then the entries from the **axisDisplay** (100810) parameter are used. The top-most entry corresponds to the top-most position. Up to 24 entries with the parameters ■ axisKey ■ name ■ rule

axisKey 100811.[Index].01501

Key name of the axis

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplayRef ▶ [Index] ▶ axisKey
Input:	Select the key name of the axis for which this display setting is valid. The key names of the axes are taken from the configuration object **CfgAxis** and displayed as a selection menu.

name 100811.[Index].01502

Axis designation

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplayRef ▶ [Index] ▶ name
Input:	max. 2 Characters

Define the axis designation that, as an alternative to the key name from **CfgAxis** is used for the display. If the parameter is not set, then the TNC7 displays the key name.

rule		100811. [Index].01503
Display rule for the axis		
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplayRef ▶ [Index] ▶ rule	
Input:	Specifies the condition for displaying the axis. **ShowAlways** Axis is always shown. The display location remains reserved even if no values for the axis can be displayed, for example if the axis is not contained in the current kinematic model. **IfKinem** Axis is shown only if it is used as an axis or a spindle in the active kinematic model. **IfKinemAxis** Axis only shown if used as axis in the active kinematics model. **IfNotKinemAxis** The axis is only shown if it is not used as an axis in the active kinematics model (e.g. as spindle). **Never** The axis is not shown.	

positionWinDisplay		100803
Type of position display in the position window		
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ positionWinDisplay	
Input:	Position display in the position window (positions display 1): **NOML.** Nominal position **ACTL** Actual position **REF ACTL** Actual position referenced to the machine datum **REF NOML** Nominal position referenced to the machine datum **LAG** Following error (servo lag) **ACTDST** Distance-to-go in the input system **REFDST** Distance-to-go in the machine system **M118**	

Traverse paths that were carried out with handwheel superimpositioning (M118)

statusWinDisplay	100804
Type of position display in the Status workspace	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ statusWinDisplay
Input:	Position display in the status window (position display 2): **NOML.** Nominal position **ACTL** Actual position **REF ACTL** Actual position referenced to the machine datum **REF NOML** Nominal position referenced to the machine datum **LAG** Following error (servo lag) **ACTDST** Distance-to-go in the input system **REFDST** Distance-to-go in the machine system **M118** Traverse paths that were carried out with handwheel superimpositioning (M118)

decimalCharacter	100805
Definition of the decimal separator for the position display	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ decimalCharacter
Input:	**"."** **","**
iTNC 530:	7280

axisFeedDisplay	100806
Display of the feed rate in the applications of the **Manual** operating mode	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisFeedDisplay
Input:	**at axis key** Display of the feed rate only if an axis direction key is pressed. The axis-specific feed rate from the machine parameter CfgFeedLimits/**manualFeed** (400304) is shown. **always minimum** Display of the feed rate for all axes, including before an axis direction key is pressed (lowest value from CfgFeedLimits/**MP_manualFeed**).

iTNC 530:	7270

spindleDisplay 100807

Display of spindle position in the position display

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ spindleDisplay
Input:	**during closed loop** Display of spindle position only if the spindle is servo-controlled **during closed loop and M5** Display of spindle position if the spindle is servo-controlled and an M5 is pending **during closed loop or M5 or tapping** Display of spindle position if the spindle is servo-controlled or if an M5 is pending, or during a tapping operation

hidePresetTable 100808

Disable the **PRESET MANAGEMENT** soft key

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ hidePresetTable
Input:	**TRUE** Access to the preset table is locked; the soft key is dimmed **FALSE** The preset table can be accessed via soft key

displayFont 100812

Font size for program display in the operating modes Program Run Full Sequence, Program Run Single Block, and Positioning with Manual Data Input.

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ displayFont
Input:	**FONT_APPLICATION_SMALL** Small font size. Same font size as in the Programming and Test Run operating modes. **FONT_APPLICATION_MEDIUM** Big font size.

iconPrioList 100813

Sequence of icons in the display

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ iconPrioList
Input:	**BASIC_ROT** **ROT_3D** **TCPM** **ACC** **TURNING**

AFC

S_PULSE

MIRROR

GPS

RADCORR

PARAXCOMP

MON_FS_OVR

compatibilityBits	100815
Settings for display behavior	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ compatibilityBits
Input:	Bit ■ 0: in the small PLC window with half the width and without a bar graph, the characters are always shown in the small font size. ■ 1: in the small PLC window with half the width and with a bar graph, the characters are always shown in the large font size.

axesGridDisplay	100816
Axes as list or group in the position display.	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axesGridDisplay
Input:	The parameter specifies whether the axes in the position display are shown as a list or as a two-column grid. Possible settings: 0 to **0** Axis display as list (default) **Quantity (n)** Axis display as two-column grid with groups of n x 2 axes
iTNC 530:	7270

CfgPosDisplayPace	101000
Display step for the individual axes	
Path:	System ▶ DisplaySettings ▶ CfgPosDisplayPace
Data object:	

displayPace	101001
Display step for position display in [mm] or [°]	
Path:	System ▶ DisplaySettings ▶ CfgPosDisplayPace ▶ [Key name of the axis] ▶ displayPace
Input:	**0.1** **0.05**

	0.01
	0.005
	0.001
	0.0005
	0.0001
	0.00005
	0.00001
	0.000005
	0.000001
iTNC 530:	7290.0-8

displayPaceInch 101002

Display step for position display in [inch]

Path:	System ▶ DisplaySettings ▶ CfgPosDisplayPace ▶ [Key name of the axis] ▶ displayPaceInch
Input:	**0.005** **0.001** **0.0005** **0.0001** **0.00005** **0.00001** **0.000005** **0.000001**
iTNC 530:	7290.0-8

CfgUnitOfMeasure 101100

Definition of unit of measure in effect for display

Path:	System ▶ DisplaySettings ▶ CfgUnitOfMeasure
Data object:	

unitOfMeasure 101101

Unit of measure for display and user interface

Path:	System ▶ DisplaySettings ▶ CfgUnitOfMeasure ▶ unitOfMeasure
Input:	**metric** Metric measurement system **inch** Inches

CfgProgramMode 101200

Format of the NC programs and cycle display

Path:	System ▶ DisplaySettings ▶ CfgProgramMode
Data object:	

programInputMode 101201

MDI: Program entry in HEIDENHAIN Klartext format or ISO format

Path:	System ▶ DisplaySettings ▶ CfgProgramMode ▶ programInputMode
Input:	**HEIDENHAIN** Program entry with HEIDENHAIN Klartext **ISO** Program entry according to ISO

CfgDisplayLanguage 101300

Definition of the NC and PLC conversational language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage
Data object:	

ncLanguage 101301

NC conversational language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ ncLanguage
Input:	**ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH** **PORTUGUESE** **SWEDISH** **DANISH** **FINNISH** **DUTCH** **POLISH** **HUNGARIAN** **RUSSIAN** **CHINESE** **CHINESE_TRAD** **SLOVENIAN** **KOREAN** **NORWEGIAN** **ROMANIAN**

	SLOVAK
	TURKISH
iTNC 530:	7230.0

applyCfgLanguage 101305

Load the language of the NC control

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ applyCfgLanguage
Input:	When booting, the control checks whether the language settings of the operating system and the NC are the same. If the settings differ, the NC applies the language setting of the operating system. If the language defined in the machine parameters of the NC is to be used, then you must the set the parameter applyCfgLanguage to TRUE.

plcDialogLanguage 101302

PLC conversational language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ plcDialogLanguage
Input:	**ENGLISH**
	GERMAN
	CZECH
	FRENCH
	ITALIAN
	SPANISH
	PORTUGUESE
	SWEDISH
	DANISH
	FINNISH
	DUTCH
	POLISH
	HUNGARIAN
	RUSSIAN
	CHINESE
	CHINESE_TRAD
	SLOVENIAN
	KOREAN
	NORWEGIAN
	ROMANIAN
	SLOVAK
	TURKISH

iTNC 530:	7230.1

plcErrorLanguage 101303

PLC error message language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ plcErrorLanguage
Input:	**ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH** **PORTUGUESE** **SWEDISH** **DANISH** **FINNISH** **DUTCH** **POLISH** **HUNGARIAN** **RUSSIAN** **CHINESE** **CHINESE_TRAD** **SLOVENIAN** **KOREAN** **NORWEGIAN** **ROMANIAN** **SLOVAK** **TURKISH**
iTNC 530:	7230.2

helpLanguage 101304

Language for online help

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ helpLanguage
Input:	**ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH**

	PORTUGUESE
	SWEDISH
	DANISH
	FINNISH
	DUTCH
	POLISH
	HUNGARIAN
	RUSSIAN
	CHINESE
	CHINESE_TRAD
	SLOVENIAN
	KOREAN
	NORWEGIAN
	ROMANIAN
	SLOVAK
	TURKISH
iTNC 530:	7230.3

CfgStartupData 101500

Behavior during control startup

Path:	System ▶ DisplaySettings ▶ CfgStartupData
Data object:	

powerInterruptMsg 101501

Acknowledge the **Power interrupted** message

Path:	System ▶ DisplaySettings ▶ CfgStartupData ▶ powerInterruptMsg
Input:	**TRUE** Start-up is only continued after the message has been acknowledged. **FALSE** The **Power interrupted** message does not appear

opMode 101503

Operating mode that is switched to when the control has fully booted

Path:	System ▶ DisplaySettings ▶ CfgStartupData ▶ opMode
Input:	Enter here the GUI designator of the desired operating mode. See the Technical Manual for an overview of the permissible GUI designators. max. 500 Characters

subOpMode 101504

Submode to be activated for the operating mode entered in 'opMode'

Path:	System ▶ DisplaySettings ▶ CfgStartupData ▶ subOpMode
Input:	Enter here the GUI designator of the desired operating submode. See the Technical Manual for an overview of the permissible GUI designators. max. 500 Characters

CfgClockView 120600

Display mode for time of day

Path:	System ▶ DisplaySettings ▶ CfgClockView
Data object:	

displayMode 120601

Display mode for time of day on the screen

Path:	System ▶ DisplaySettings ▶ CfgClockView ▶ displayMode
Input:	**Analog** Analog clock **Digital** Digital clock **Logo** OEM logo **Analog and logo** Analog clock and OEM logo **Digital and logo** Digital clock and OEM logo **Analog on logo** Analog clock that superimposes the OEM logo **Digital on logo** Digital clock that superimposes the OEM logo

timeFormat 120602

Time format of digital clock

Path:	System ▶ DisplaySettings ▶ CfgClockView ▶ timeFormat
Input:	Possible settings: **12 h format** Time in 12 hours format **24 h format** Time in 24 hours format

CfgInfoLine 120700

Link row on/off

Path:	System ▶ DisplaySettings ▶ CfgInfoLine
Data object:	

infoLineEnabled 120701

Enable/disable info line

Path:	System ▶ DisplaySettings ▶ CfgInfoLine ▶ infoLineEnabled
Input:	**OFF** The info line is disabled **ON** The info line below the operating mode display is enabled

CfgGraphics 124200

Settings for 3D simulation graphics

Path:	System ▶ DisplaySettings ▶ CfgGraphics
Data object:	

modelType 124201

Model type of the 3D simulation graphics

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ modelType
Input:	**No Model** The model depiction is deactivated. Only the 3D line graphics are shown (lowest processor load, e.g. for fast testing of the NC program and ascertainment of program run times) **3D** Model depiction for complex operations (highest processor load, e.g. for turning or undercuts) **2.5D** Model depiction for 3-axis operations (medium processor load)

modelQuality 124202

Model quality of the 3D simulation graphics

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ modelQuality
Input:	**very high** Very high model quality, the production result can be precisely judged. This setting requires the highest computing power. Block numbers and block end points can only be displayed in the 3D line graphics with this setting. **high** High model quality **medium** Medium model quality **low** Low model quality

clearPathAtBlk 124203

Reset tool paths for new BLK FORM

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ clearPathAtBlk
Input:	**ON** With a new BLK FORM in the Test Run graphic, the tool paths are reset **OFF** With a new BLK FORM in the Test Run graphic, the tool paths are not reset

extendedDiagnosis 124204

Write graphics journal files after restart

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ modelType
Input:	Activate diagnostic information for HEIDENHAIN (journal files) for the analysis of graphics problems. **OFF** Do not create journal files (default). **ON** Create journal files.

CfgPositionDisplay 124500

Settings for the digital readout

Path:	System ▶ DisplaySettings ▶ CfgPositionDisplay
Data object:	

progToolCallDL 124501

Position display with TOOL CALL DL

Path:	System ▶ DisplaySettings ▶ CfgPositionDisplay ▶ progToolCallDL
Input:	**As Tool Length** The oversize DL programmed in the TOOL CALL block is taken into account as part of the tool length in the nominal position display. **As Workpiece Oversize** The programmed oversize DL in the TOOL CALL block is not taken into account in the nominal position display. It therefore has the effect of a workpiece oversize.

CfgTableEditor 125300

Table editor configuration

Path:	System ▶ TableSettings ▶ CfgTableEditor
Data object:	Specifies properties and settings for the table editor.

deleteLoadedTool 125301

Behavior when deleting tools from the pocket table

Path:	System ▶ TableSettings ▶ CfgTableEditor ▶ deleteLoadedTool
Input:	Possible settings: **DISABLED** Tool deletion is not possible **WITH_WARNING** Tool deletion is possible; Note must be confirmed **WITHOUT_WARNING** Tool deletion is possible without confirmation
iTNC 530:	7263 Bit4, 7263 Bit5

indexToolDelete 125302

Behavior when deleting a tool's index entries

Path:	System ▶ TableSettings ▶ CfgTableEditor ▶ indexToolDelete
Input:	Possible settings: **ALWAYS_ALLOWED** Deletion of index entries is always possible **TOOL_RULES** Behavior depends on the setting of the parameter delete-LoadedTool
iTNC 530:	7263 Bit6

showResetColumnT 125303

Show the **RESET COLUMN T** soft key

Path:	System ▶ TableSettings ▶ CfgTableEditor ▶ showResetColumnT
Input:	The parameter specifies whether the **RESET COLUMN T** soft key is to be offered when a pocket table is open in the table editor. **TRUE** The soft key is displayed. All tools can be deleted from the tool memory by the user. **FALSE** The soft key is not displayed.
iTNC 530:	7263 Bit3

CfgDisplayCoordSys 127500

Setting the coordinate systems for the display

Path:	System ▶ DisplaySettings ▶ CfgDisplayCoordSys
Data object:	

transDatumCoordSys 127501

Coordinate system for the datum shift

Path:	System ▶ DisplaySettings ▶ CfgDisplayCoordSys ▶ transDatumCoordSys
Input:	The parameter specifies the coordinate system in which the datum shift is displayed. **WorkplaneSystem** Datum is displayed in the system of the tilted plane (WPL-CS) **WorkpieceSystem** Datum is displayed in the workpiece coordinate system (W-CS)

CfgGlobalSettings 128700

GPS display settings

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings
Data object:	

enableOffset 128702

Show offset in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableOffset
Input:	**OFF** Offset is not displayed **ON** Offset is displayed

enableBasicRot 128703

Show an additive basic rotation in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableBasicRot
Input:	**OFF** Additive basic rotation is not displayed **ON** Additive basic rotation is displayed

enableShiftWCS 128704

Show shift of W-CS in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableShiftWCS
Input:	**OFF** Shift of W-CS (workpiece coordinate system) is not displayed **ON**

Shift of W-CS (workpiece coordinate system) is displayed

enableMirror		128712
Show mirroring in the GPS dialog		
Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableMirror	
Input:	**OFF** Mirroring is not displayed **ON** Mirroring is displayed	

enableShiftMWCS		128711
Show shift of mW-CS in the GPS dialog		
Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableShiftMWCS	
Input:	**OFF** Shift of mW-CS (modified workpiece coordinate system) is not displayed **ON** Shift of the mW-CS (modified workpiece coordinate system) is displayed	

enableRotation		128707
Show rotation in the GPS dialog		
Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableRotation	
Input:	**OFF** Rotation is not displayed **ON** Rotation is displayed	

enableFeed		128708
Show feed rate in the GPS dialog		
Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableFeed	
Input:	**OFF** Feed rate is not displayed **ON** Freed rate is displayed	

enableHwMCS		128709
M-CS coordinate system is selectable		
Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwMCS	

Input:	**OFF** Coordinate system M-CS (machine coordinate system) cannot be selected **ON** Coordinate system M-CS (machine coordinate system) can be selected

enableHwWCS 128710

W-CS coordinate system is selectable

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwWCS
Input:	**OFF** Coordinate system W-CS (workpiece coordinate system) cannot be selected **ON** Coordinate system W-CS (workpiece coordinate system) can be selected

enableHwMWCS 128711

mW-CS coordinate system is selectable

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwMWCS
Input:	**OFF** Coordinate system mW-CS (modified workpiece coordinate system) cannot be selected **ON** Coordinate system mW-CS (modified workpiece coordinate system) can be selected

enableHwWPLCS 128712

WPL-CS coordinate system is selectable

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwWPLCS
Input:	**OFF** Coordinate system WPL-CS (working plane coordinate system) cannot be selected **ON** Coordinate system WPL-CS (working plane coordinate system) can be selected

enableHwAxisU 128713

U axis can be selected

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwAxisU
Input:	**OFF** U axis cannot be selected

ON
U axis can be selected

enableHwAxisV		128714
V axis can be selected		
Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwAxisV	
Input:	**OFF** V axis cannot be selected **ON** V axis can be selected	

enableHwAxisW		128715
W axis can be selected		
Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwAxisW	
Input:	**OFF** W axis cannot be selected **ON** W axis can be selected	

CfgRemoteDesktop		133500
Settings for Remote Desktop connections		
Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop	
Data object:		

connections		133501
List of Remote Desktop connections to be displayed		
Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ connections	
Input:	Enter here the name of a RemoteFX connection from Remote Desktop Manager. max. 80 Characters	

autoConnect		133505
Start connection automatically		
Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ autoConnect	
Input:	**TRUE** Automatically connect when control boots **FALSE** Do not start connection automatically.	

title		133502

Name of the OEM operating mode

	Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ title
	Input:	Specifies the name of the OEM operating mode for display on the TNC and in the information bar.

dialogRes 133502.00501

Name of a text

	Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ title ▶ dialogRes
	Input:	If the text is not meant to be language-sensitive, then it must be available with this name in a text resource file. In this case, enter the text at the “text” attribute. max. 40 Characters

text 133502.00502

Language-sensitive text

	Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ title ▶ text
	Input:	This text is loaded from a text resource file and should not be changed here. If the text is not language-sensitive, then you must directly provide the text here. In this case, do not enter anything at the “dialogRes” attribute. max. 60 Characters

icon 133503

Path/name for optional icon graphic file

	Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ icon
	Input:	max. 260 Characters

locations 133504

List with positions where this Remote Desktop connection is displayed

	Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ locations
	Input:	

opMode 133504.[Index].133401

Operating mode

	Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ locations ▶ [Index] ▶ opMode
	Input:	max. 80 Characters

subOpMode 133504.[Index].133402

Optional submode for the operating mode specified in 'opMode'

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ locations ▶ [Index] ▶ subOpMode
Input:	max. 80 Characters

PalletSettings

CfgPalletBehaviour 202100

Behavior of the pallet control cycle

Path:	System ▶ PalletSettings ▶ CfgPalletBehaviour
Data object:	

failedCheckReact 202106

Specify reaction to program check and tool check

Path:	System ▶ PalletSettings ▶ CfgPalletBehaviour ▶ failedCheckReact
Input:	**Never** No checking for faulty program or tool calls. **OnFailedPgmCheck** Check for faulty program calls. **OnFailedToolCheck** Check for faulty tool calls.

failedCheckImpact 202107

Specify effect of program check or tool check

Path:	System ▶ PalletSettings ▶ CfgPalletBehaviour ▶ failedCheckImpact
Input:	**SkipPGM** Skip faulty programs. **SkipFIX** Skip fixture setups that contain faulty programs. **SkipPAL** Skip pallets that contain faulty programs.

ProbeSettings

CfgTT 122700

Configuration of the tool calibration

Path:	System ▶ ProbeSettings ▶ CfgTT
Data object:	

spindleOrientMode 122704

M function for spindle orientation

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ spindleOrientMode
Input:	-1 to 999 ■ **-1** Spindle orientation directly by NC ■ **0** Function inactive ■ **1 to 999** Number of the M function for spindle orientation by the PLC
iTNC 530:	MP6560

probingRoutine 122705

Probing routine

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingRoutine
Input:	**MultiDirections** The probe contact is probed from several directions. **SingleDirection** The probe contact is probed from one direction.
iTNC 530:	6500 Bit 8

probingDirRadial 122706

Probing direction for tool radius measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingDirRadial
Input:	**X_Positive** **Y_Positive** **X_Negative** **Y_Negative** **Z_Positive** **Z_Negative**

iTNC 530:	MP6505

offsetToolAxis 122707

Distance from lower edge of tool to upper edge of stylus

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ offsetToolAxis
Input:	0.001 to 99.9999 [mm], max. 4 decimal places
iTNC 530:	MP6530

rapidFeed 122708

Rapid traverse in probing cycle for TT tool touch probe

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ rapidFeed
Input:	10 to 300000
iTNC 530:	MP6550

probingFeed 122709

Probing feed rate for tool measurement with non-rotating tool

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingFeed
Input:	1 to 3000
iTNC 530:	6520

probingFeedCalc 122710

Calculation of the probing feed rate

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingFeedCalc
Input:	**ConstantTolerance** Calculation of the probing feed rate with constant tolerance **VariableTolerance** Calculation of the probing feed rate with variable tolerance **ConstantFeed** Constant probing feed rate
iTNC 530:	6507

spindleSpeedCalc 122711

Speed determination method

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ spindleSpeedCalc
Input:	**Automatic** Automatically determine speed **MinSpindleSpeed** Always use minimum spindle speed

iTNC 530:	6500 Bit4

maxPeriphSpeedMeas 122712

Maximum permissible surface speed of the tool edge for radius measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ maxPeriphSpeedMeas
Input:	1 to 129 [m/min], max. 4 decimal places
iTNC 530:	6570

maxSpeed 122714

Maximum permissible speed during tool measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ maxSpeed
Input:	0 to 1000
iTNC 530:	6572

measureTolerance1 122715

Maximum permissible measuring error for tool measurement with rotating tool (first measurement error)

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ measureTolerance1
Input:	0.001 to 0.999 [mm], max. 3 decimal places
iTNC 530:	6510.0

measureTolerance2 122716

Maximum permissible measuring error for tool measurement with rotating tool (second measurement error)

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ measureTolerance2
Input:	0.001 to 0.999 [mm], max. 3 decimal places
iTNC 530:	6510.1

stopOnCheck 122717

NC stop during tool check

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ stopOnCheck
Input:	**TRUE** If the breakage tolerance is exceeded, the NC program is stopped and the error message **Tool broken** is displayed. **FALSE** The NC program is not stopped when the breakage tolerance is exceeded

iTNC 530:	6500 Bit5

stopOnMeasurement 122718

NC stop during tool measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ stopOnMeasurement
Input:	**TRUE** If the breakage tolerance is exceeded, the NC program is stopped and the error message **Touch point inaccessible** is displayed **FALSE** The NC program is not stopped when the breakage tolerance is exceeded
iTNC 530:	6500 Bit6

adaptToolTable 122719

Change the tool table during tool check and tool measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ adaptToolTable
Input:	**AdaptNever** The tool table is not changed after tool check and tool measurement. **AdaptOnBoth** The tool table is changed after tool check and tool measurement. **AdaptOnMeasure** The tool table is changed after tool measurement.
iTNC 530:	6500 Bit11

CfgTTRoundStylus 114200

Configuration of a round stylus

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus
Data object:	

centerPos 114201

Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus ▶ [Key name of the TT] ▶ centerPos
Input:	-99999.9999 to 99999.9999 [mm], max. 4 decimal places [0]: X coordinate [1]: Y coordinate [2]: Z coordinate

iTNC 530:	6580, 6581, 6582

safetyDistToolAx 114203

Safety clearance around the probe contact of the TT tool touch probe for pre-positioning in the tool-axis direction

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus ▶ [Key name of the TT] ▶ safetyDistToolAx
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places
iTNC 530:	6540.0

safetyDistStylus 114204

Safety zone around the stylus for pre-positioning

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus ▶ [Key name of the TT] ▶ safetyDistStylus
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places Safety clearance in the plane perpendicular to the tool axis
iTNC 530:	6540.1

CfgTTRectStylus 114300

Configuration of a rectangular stylus

Path:	System ▶ ProbeSettings ▶ CfgTTRectStylus
Data object:	

centerPos 114313

Coordinates of the stylus center

Path:	System ▶ ProbeSettings ▶ CfgTTRectStylus ▶ [Key name of the TT] ▶ centerPos
Input:	Coordinates of the stylus center with respect to the machine datum -99999.9999 to 99999.9999 [mm], max. 4 decimal places
iTNC 530:	6580, 6581, 6582

safetyDistToolAx 114317

Set-up clearance above the stylus for pre-positioning

Path:	System ▶ ProbeSettings ▶ CfgTTRectStylus ▶ [Key name of the TT] ▶ safetyDistToolAx
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places Safety clearance in tool axis direction
iTNC 530:	6540.0

safetyDistStylus 114318

Safety zone around the stylus for pre-positioning

Path:	System ▶ ProbeSettings ▶ CfgTTRectStylus ▶ [Key name of the TT] ▶ safetyDistStylus
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places
iTNC 530:	6540.1

ChannelSettings

CfgActivateKinem	204000
Active kinematics	
Path:	Channels ▶ ChannelSettings ▶ CfgActivateKinem
Data object:	

kinemToActivate	204001
Kinematics to be activated / active kinematics	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgActivateKinem ▶ kinemToActivate
Input:	max. 18 Characters Key names from channels/kinematics/**CfgKinComposModel**. Select the key name of the kinematic model to be activated. You can also read the currently active kinematic model from this machine parameter.

kinemAtStartup	204002
The kinematics to be activated during control start-up	
Path:	Channels ▶ ChannelSettings ▶ CfgActivateKinem ▶ [Key name of the machining channel] ▶ kinemAtStartup
Input:	max. 18 Characters Enter here the key name of a default kinematic model (from **CfgKinComposModel**), that is activated during every control start-up (independently of which key name is entered in the machine parameter **kinemToActivate** (204001)).
iTNC 530:	7506

CfgNcPgmBehaviour	200800
Specify the behavior of the NC program.	
Path:	Channels ▶ ChannelSettings ▶ CfgNcPgmBehaviour
Data object:	

operatingTimeReset	200801
Reset the machining time when program starts.	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgNcPgmBehaviour ▶ operatingTimeReset
Input:	**TRUE** The machining time is reset at each program start. **FALSE**

The machining time is totaled.

plcSignalCycle		200803
PLC signal for the number of the pending machining cycle		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgNcPgmBehaviour ▶ plcSignalCycle
	Input:	max. 500 Characters Name or number of a PLC word marker

CfgGeoTolerance		200900
Geometry tolerances		
	Path:	Channels ▶ ChannelSettings ▶ CfgGeoTolerance
	Data object:	

circleDeviation		200901
Permissible deviation of the radius		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoTolerance ▶ circleDeviation
	Input:	0.0001 to 0.016 [mm], max. 4 decimal places Enter the permissible deviation of the radius between the end point and starting point of the arc.
	iTNC 530:	7431

threadTolerance		200902
Permissible deviation in successive threads		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoTolerance ▶ threadTolerance
	Input:	0.0001 to 999.9999 [mm], max. 9 decimal places Permissible deviation of the dynamically smoothed contour from the programmed thread contour.

moveBack		200903
Reserve for retraction movements		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoTolerance ▶ moveBack
	Input:	0.0001 to 10 [mm], max. 9 decimal places With this parameter you specify how far before a limit switch or a collision object a retraction movement should end.

CfgGeoCycle		201000

Configuration of the fixed cycles

Path:	Channels ▶ ChannelSettings ▶ CfgGeoCycle
Data object:	

pocketOverlap 201001

Overlap factor for pocket milling

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ pocketOverlap
Input:	0.001 to 1.414, max. 3 decimal places
iTNC 530:	7430

posAfterContPocket 201007

Traverse after machining the contour pocket

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ posAfterContPocket
Input:	**PosBeforeMachining** Move to the position from which the SL cycle was started. **ToolAxClearanceHeight** Move the tool axis to clearance height.
iTNC 530:	7420 Bit 4

displaySpindleErr 201002

Display the **Spindle is not rotating** error message if M3/M4 is not active

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ displaySpindleErr
Input:	**on** The error message is displayed **off** The error message is not displayed
iTNC 530:	7441

displayDepthErr 201003

Display the **Check the depth sign** error message

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ displayDepthErr
Input:	**on** Error message is displayed **off** Error message is not displayed

iTNC 530:	7441

apprDepCylWall 201004

Behavior when moving to wall of slot in the cylinder surface

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► apprDepCylWall
Input:	Defines the behavior for cutter movements to the wall of a slot in the cylinder surface when machining the slot with a milling cutter whose diameter is less than the slot diameter (e.g. Cycle 28). **LineNormal** The slot wall is approached and departed linearly. **CircleTangential** The slot wall is approached and departed tangentially; at the beginning and end of the slot a rounding arc with a diameter equal to the slot width is inserted.
iTNC 530:	7680 Bit 12

mStrobeOrient 201005

M function for spindle orientation in machining cycles

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► mStrobeOrient
Input:	-1 to 999 -1: Spindle orientation directly through the NC 0: Function not active 1 to 999: Number of the M function for spindle orientation through the PLC.
iTNC 530:	7442

suppressPlungeErr 201006

Do not show 'Plunging type is not possible' error message

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► suppressPlungeErr
Input:	**on** Error message is not displayed **off** Error message is displayed

restoreCoolant 201008

Behavior of M7 and M8 with Cycles 202 and 204

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► restoreCoolant

Input:	**TRUE** At the end of Cycles 202 and 204, the status of M7 and M8 is restored to that before the cycle call. **FALSE** At the end of Cycles 202 and 204, the status of M7 and M8 is not restored automatically.
iTNC 530:	7682

facMinFeedTurnSMAX 201009

Automatic feed rate reduction after attaining SMAX

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► facMinFeedTurnSMAX
Input:	1 to 100 [%], max. 1 decimal places If the maximum spindle speed SMAX is reached, the turning operation can no longer maintain the constant surface speed (VCONST:ON). The parameter determines whether the feed should be automatically reduced from this point to the center of rotation. Possible settings: ■ Factor = 100% (default value): Feed rate reduction deactivated. The feed rate from the turning cycle is used. ■ 0 < factor < 100%: Feed rate reduction is activated. The minimum feed rate Fmin is: Fmin = feed rate from turning cycle * factor

suppressResMatlWar 201010

Do not show "Residual material" warning

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► suppressResMatlWar
Input:	**Never** The "Residual material due to cutter geometry" warning is never suppressed **NCOnly** The “Residual material due to cutter geometry” warning is suppressed only in the Machine operating modes. **Always** The "Residual material due to cutter geometry" warning is always suppressed.

CfgStretchFilter 201100

Geometry filter for filtering out linear elements

Path:	Channels ► ChannelSettings ► CfgStretchFilter

Data object:

filterType 201101

Type of stretch filter

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgStretchFilter ▶ filterType
Input:	**Off** The filter is switched off. **ShortCut** Individual points on the polygon are omitted. If the connecting line from the middle point to the previous point or to the next point of three subsequent points on a polygon is within the tolerance band, the middle point will be omitted. **Average** The geometry filter smooths corners. This method moves the contour points in such a way that changes in direction are less distinct.

tolerance 201102

Maximum distance of the filtered to the unfiltered contour

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgStretchFilter ▶ tolerance
Input:	0 to 10 [mm], max. 5 decimal places Points that lie within this tolerance on the resulting new path will be filtered out. **0**: Stretch filter is off

maxLength 201103

Maximum length of the distance resulting from filtering

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgStretchFilter ▶ maxLength
Input:	0 to 1000 [mm], max. 3 decimal places **0**: Stretch filter is off

CfgThreadSpindle 113600

Special spindle parameters for threads

Path:	Channels ▶ ChannelSettings ▶ CfgThreadSpindle
Data object:	

sourceOverride 113603

Effective override potentiometer for feed rate during thread cutting

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ sourceOverride
Input:	The adjusted potentiometer is effective during thread cutting for shaft speed and feed rate. **FeedPotentiometer** (previous behavior of the TNC 640) During thread cutting, the potentiometer is effective for the feed rate knob. The potentiometer for the spindle speed knob is not active. **SpindlePotentiometer** (iTNC 530-compatible setting) During thread cutting, the potentiometer is effective for the spindle speed knob. The potentiometer for the feed rate override is disabled.

thrdWaitingTime 113601

Waiting time at reversal point in thread base

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ thrdWaitingTime
Input:	0 to 1 000 [s], max. 9 decimal places The spindle stops for this time at the bottom of the thread before starting again in the opposite direction of rotation.
iTNC 530:	7120.0

thrdPreSwitchTime 113602

Advanced switching time of spindle

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ thrdPreSwitchTime
Input:	0 to 1 000 [s], max. 9 decimal places The spindle is stopped at this time before reaching the bottom of the thread.
iTNC 530:	7120.1

limitSpindleSpeed 113604

Limit of spindle speed with Cycles 17, 207 and 18

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ limitSpindleSpeed
Input:	**TRUE** Spindle speed is limited so that it runs with constant speed approx. 1/3 of the time **FALSE** Limit not active
iTNC 530:	7160, Bit1

CfgEditorSettings

CfgEditorSettings		105400
Settings for the NC editor		
Path:	System ▶ EditorSettings ▶ CfgEditorSettings	
Data object:		

createBackup		105401
Generate a backup file *.bak		
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ createBackup	
Input:	**TRUE** After you have edited a file, a backup file *.bak is automatically created before you save the file and exit the NC editor. **FALSE** No backup file *.bak is created. Select this setting if you do not need any backup files and want to save memory space.	

deleteBack		105402
Behavior of the cursor after deletion of lines		
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ deleteBack	
Input:	**TRUE** Behavior as with iTNC 530, the cursor is on the previous line **FALSE** The cursor is on the next line	

lineBreak		105404
Line break on NC blocks with more than one line		
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ lineBreak	
Input:	**ALL** Always break and display lines completely (multiline) **ACT** Only display the selected NC block completely (multiline) **NO** Only display all lines when the selected NC block is edited	
iTNC 530:	7281.0	

stdTNChelp		105405
Activate help graphics when entering cycle data		
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ stdTNChelp	
Input:	**TRUE**	

Behavior as with iTNC 530: the help graphics are displayed automatically during cycle entry.

FALSE

The help graphics have to be called via the **CYCLE HELP ON/OFF** soft key.

warningAtDEL 105407

Confirmation request when deleting an NC block.

Path:	System ► EditorSettings ► CfgEditorSettings ► warningAtDEL
Input:	**TRUE** The confirmation request is displayed and must be confirmed by pressing DEL again. **FALSE** iTNC 530 behavior: The NC block is deleted without any request for confirmation.
iTNC 530:	7246

maxLineGeoSearch 105408

Line number up to which a test of the NC program is to be run.

Path:	System ► EditorSettings ► CfgEditorSettings ► maxLineGeoSearch
Input:	The available value range depends on the performance of the control. For the TNC7, you can enter a value between 100 and 100 000. If the parameter is not part of the configuration, the minimal value 100 becomes effective.
iTNC 530:	7229

blockIncrement 105409

ISO programming: Block number increment

Path:	System ► EditorSettings ► CfgEditorSettings ► blockIncrement
Input:	0 to 250
iTNC 530:	7220

useProgAxes 105410

Specify programmable axes

Path:	System ► EditorSettings ► CfgEditorSettings ► useProgAxes
Input:	**TRUE** Use the axis configuration defined in the CfgChannelAxes/**progAxis** parameter (200301). On machines with traverse range switchover, the editor offers all axes that are included in at least one kinematic model of the machine. **FALSE**

Use the default axis configuration XYZABCUVW.

enableStraightCut	105411
Allow or lock paraxial positioning blocks	
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ enableStraightCut
Input:	**TRUE** Paraxial positioning blocks are allowed. When an orange axis key is pressed, and in ISO when G07 is programmed, a paraxial positioning block is generated. **FALSE** Paraxial positioning blocks are locked. When an orange axis key is pressed, the TNC7 generates a straight-line interpolation (L block) instead of a paraxial positioning block.
iTNC 530:	7246

noParaxMode	105413
Hide **FUNCTION PARAXCOMP/PARAXMODE**	
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ noParaxMode
Input:	Use **noParaxMode** (105413) to hide the **FUNCTION PARAXCOMP** and **FUNCTION PARAXMODE** functions. **FALSE** The functions are displayed **TRUE** The functions are not displayed If the optional machine parameter does not exist in the configuration, the system behaves as if it were set to **FALSE**.

CfgPgmMgt

CfgPgmMgt	122100
Settings for the file management	
Path:	System ▶ ProgramManager ▶ CfgPgmMgt
Data object:	

dependentFiles	122101
Display of dependent files	
Path:	System ▶ ProgramManager ▶ CfgPgmMgt ▶ dependentFiles
Input:	**AUTOMATIC** Dependent files are not displayed **MANUAL** Dependent files are displayed

CfgProgramCheck

CfgProgramCheck		129800
Settings for tool-usage files		
	Path:	System ▶ ToolSettings ▶ CfgProgramCheck
	Data object:	

autoCheckTimeOut		129803
Timeout for creation of tool-usage files		
	Path:	System ▶ ToolSettings ▶ CfgProgramCheck ▶ autoCheckTimeOut
	Input:	Automatic creation of the tool-usage file is aborted if this time is exceeded. 1 to 500

autoCheckPrg		129801
Create tool-usage file for NC program		
	Path:	System ▶ ToolSettings ▶ CfgProgramCheck ▶ autoCheckPrg
	Input:	**NoAutoCreate** No tool-usage list will be generated upon selection of a program **OnProgSelectionIfNotExist** A tool-usage list will be generated upon program selection if the list does not already exist **OnProgSelectionIfNecessary** A tool-usage list will be generated upon program selection if the list does not already exist or if it contains obsolete data **OnProgSelectionAndModify** A tool usage list will be generated upon program selection if the list does not already exist, if it contains obsolete data, or if the NC program is modified afterwards using an editor

autoCheckPal		129802
Create pallet-usage files		
	Path:	System ▶ ToolSettings ▶ CfgProgramCheck ▶ autoCheckPal
	Input:	**NoAutoCreate** No tool-usage files will be generated upon pallet selection **OnProgSelectionIfNotExist** Upon pallet selection, those tool-usage lists that do not already exist will be generated **OnProgSelectionIfNecessary** Upon pallet selection, those tool-usage lists that do not already exist or that contain obsolete data will be generated **OnProgSelectionAndModify**

Upon pallet selection, tool-usage lists will be generated that do not already exist, that contain obsolete data, or whose NC programs are modified via the editor

CfgUserPath

CfgUserPath 102200

Paths for the end user

Path:	System ▶ Paths ▶ CfgUserPath
Data object:	

ncDir 102201

List of drives and/or directories

Path:	System ▶ Paths ▶ CfgUserPath ▶ ncDir
Input:	max. 260 Characters This parameter is available only on the Windows programming stations of the TNC7. The parameter is not evaluated on a programming station with virtualization software (VBox) or on the TNC target system. The drives and/or directories entered here are visible in the file manager, provided that you have the required access rights. These paths may contain NC programs or tables. Possible entries are, for example: floppy-disk, HDR and CFR directories as well as network drives.

fn16DefaultPath 102202

Default output path for the **FN16: F-PRINT** function in the Program Run operating modes

Path:	System ▶ Paths ▶ CfgUserPath ▶ fn16DefaultPath
Input:	max. 260 Characters Select the folder in the dialog window and confirm it with the **SELECT** soft key Default path for output with **FN 16: F-PRINT**. If no path is defined for the FN 16 function in the NC program, the output destination is in the directory specified here.

fn16DefaultPathSim 102203

Default output path for the **FN16: F-PRINT** function in the Programming and Test Run operating modes

Path:	System ▶ Paths ▶ CfgUserPath ▶ fn16DefaultPathSim
Input:	max. 260 Characters Select the folder in the dialog window and confirm it with the **SELECT** soft key Default path for output with **FN 16: F-PRINT**. If no path is defined for the FN 16 function in the NC program, the output destination is in the directory specified here.

serialInterfaceRS232

CfgSerialPorts 106600

Data record belonging to the serial port

Path:	System ▶ Network ▶ Serial ▶ CfgSerialPorts
Data object:	

activeRs232 106601

Enable the RS-232 interface in the program manager

Path:	System ▶ Network ▶ Serial ▶ CfgSerialPorts ▶ activeRs232
Input:	**TRUE** The RS-232 interface is enabled in the program manager and shown as a drive icon (**RS232:**). **FALSE** The RS-232 interface cannot be accessed via the program manager.

baudRateLsv2 106606

Data transfer rate for LSV2 communication in baud

Path:	System ▶ Network ▶ Serial ▶ CfgSerialPorts ▶ baudRateLsv2
Input:	Use a selection menu to define the transfer rate for the LSV2 communication. Minimum value is 110 baud, maximum value 115200 baud. **BAUD_110** **BAUD_150** **BAUD_300** **BAUD_600** **BAUD_1200** **BAUD_2400** **BAUD_4800** **BAUD_9600** **BAUD_19200** **BAUD_38400** **BAUD_57600** **BAUD_115200**

CfgSerialInterface 106700

Definition of data records for the serial ports

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface

Data object:

baudRate		106701
Data transfer rate for communication in baud		
	Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ baudRate
	Input:	Use a selection menu to define the transfer rate for the data transmission. Minimum value is 110 baud, maximum value 115200 baud. **BAUD_110** **BAUD_150** **BAUD_300** **BAUD_600** **BAUD_1200** **BAUD_2400** **BAUD_4800** **BAUD_9600** **BAUD_19200** **BAUD_38400** **BAUD_57600** **BAUD_115200**
	iTNC 530:	5040

protocol		106702
Communications protocol		
	Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ protocol
	Input:	**STANDARD** Standard data transfer. Data transferred line-by-line. **BLOCKWISE** Packet-based data transfer, ACK/NAK protocol. The control characters ACK (Acknowledge) and NAK (not Acknowledge) are used to control block-wise data transfer. **RAW_DATA** Data transferred without protocol. Transfer of characters without control characters. Protocol intended for transfer of data of the PLC.
	iTNC 530:	5030

dataBits		106703
Data bits in each transferred character		
	Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ dataBits

Input:	**7 bits** 7 data bits are transferred for each character transferred. **8 bits** 8 data bits are transferred for each character transferred.
iTNC 530:	5020 Bit0

parity 106704

Type of parity checking

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ parity
Input:	**NONE** No parity **EVEN** Even parity **ODD** Odd parity
iTNC 530:	5020 Bit4/5

stopBits 106705

Number of stop bits

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ stopBits
Input:	**1 stop bit** 1 stop bit is appended after each transferred character. **2 stop bits** 2 stop bits are appended after each transferred character.
iTNC 530:	5020 Bit6/7

flowControl 106706

Type of data-flow checking

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ flowControl
Input:	Configure here whether there is to be a data-flow check (handshake). **NONE** No data-flow check; handshake not active **RTS_CTS** Hardware handshake. Transmission stop through RTS active **XON_XOFF** Software handshake; Transfer stop by DC3 (XOFF) active
iTNC 530:	5020 Bit2/3

fileSystem 106707

File system for file operation via serial interface

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ fileSystem
Input:	**EXT** Minimum file system for external devices. Corresponds to the EXT1 and EXT2 modes of earlier TNC controls. Use these settings if you are using printers, punches, or non-HEIDENHAIN data transfer software. **FE1** Use this setting for communication with the external HEIDENHAIN FE 401 B or FE 401 floppy disk unit as of software 230626-03, or for communication with the "TNCserver" PC software from HEIDENHAIN.

bccAvoidCtrlChar 106708

Avoid control characters in the block check character (BCC)

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ bccAvoidCtrlChar
Input:	**TRUE** Ensures that the check sum does not correspond to a control character **FALSE** Function not active
iTNC 530:	5020 Bit1

rtsLow 106709

Idle state of the RTS line

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ rtsLow
Input:	**TRUE** The idle state of the RTS line is logical LOW **FALSE** The idle state of the RTS line is at logical HIGH
iTNC 530:	5020 Bit8

noEotAfterEtx 106710

Behavior after reception of an ETX control character

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ noEotAfterEtx
Input:	**TRUE** No EOT control character is sent after reception of an ETX control character. **FALSE** The control sends an EOT control character after reception of an ETX control character.

iTNC 530:	5020 Bit9

Monitoring

CfgMonUser 129400

Monitoring settings for the user

Path:	System ▶ Monitoring ▶ ComponentMonitoring ▶ CfgMonUser
Data object:	

enforceReaction 129401

The configured error reactions are enforced

Path:	System ▶ Monitoring ▶ ComponentMonitoring ▶ CfgMonUser ▶ enforceReaction
Input:	**TRUE** **FALSE**

showWarning 129402

Display warnings of monitoring tasks

Path:	System ▶ Monitoring ▶ ComponentMonitoring ▶ CfgMonUser ▶ showWarning
Input:	**TRUE** **FALSE**

CfgMonMbSection 133700

CfgMonMbSection defines monitoring tasks for a certain section of an NC program

Path:	System ▶ Monitoring ▶ ProcessMonitoring ▶ CfgMonMbSection
Data object:	

tasks 133701

List of monitoring tasks to be performed

Path:	System ▶ Monitoring ▶ ProcessMonitoring ▶ CfgMonMbSection ▶ [Key name] ▶ tasks
Input:	

CfgMachineInfo

CfgMachineInfo	131700
General information of the machine operator	
Path:	System ▶ CfgMachineInfo
Data object:	Defines general information about this machine: ■ Settable by the user of the machine ■ Can be queried (e.g., via the OPC UA NC Server)

machineNickname	131701
Custom name (nickname) of the machine	
Path:	System ▶ CfgMachineInfo ▶ machineNickname
Input:	max. 64 Characters Machine designation freely selectable by the user.

inventoryNumber	131702
Inventory number or ID	
Path:	System ▶ CfgMachineInfo ▶ inventoryNumber
Input:	max. 64 Characters Internal inventory number of the operator's machine.

image	131703
Photo or image of the machine	
Path:	System ▶ CfgMachineInfo ▶ image
Input:	max. 260 Characters Path to an image file (*.jpg or *.png).

location	131704
Machine location	
Path:	System ▶ CfgMachineInfo ▶ location
Input:	max. 64 Characters

department	131705
Department or division	
Path:	System ▶ CfgMachineInfo ▶ department
Input:	max. 64 Characters

responsibility	131706
Responsible for the machine	
Path:	System ▶ CfgMachineInfo ▶ responsibility
Input:	max. 64 Characters

Contact partner responsible for the machine, can be a person or a department.

contactEmail		131707
Contact email address		
Path:	System ▶ CfgMachineInfo ▶ contactEmail	
Input:	max. 64 Characters E-mail address of the responsible person or department.	

contactPhoneNumber		131708
Contact phone number		
Path:	System ▶ CfgMachineInfo ▶ contactPhoneNumber	
Input:	max. 32 Characters Telephone number of the responsible person or department.	

43.3 User administration roles and rights

43.3.1 List of roles

The following contents can change in the following software versions of the control:

- HEROS role names
- Unix groups
- Basic ID number

Further information: "Roles", Page 2162

Operating system roles:

Role	**Privileges**		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	■ HEROS.MountShares ■ HEROS.Printer	■ mnt ■ lp	■ 332 ■ 9
HEROS.NormalUser	Role for a normal user with limited rights on the operating system.		
	This role grants the rights of the RestrictedUser role, as well as the following rights:		
	■ HEROS.SetShares ■ HEROS.ControlFunctions	■ mntcfg ■ ctrlfct	■ 331 ■ 337

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.LegacyUser	With the **LegacyUser** role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. User administration remains active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 334
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.ReadLogs	■ logread	■ 342
	■ HEROS.SWUpdate	■ swupdate	■ 338
	■ HEROS.SetNetwork	■ netadmin	■ 333
	■ HEROS.SetTimezone	■ tz	■ 330
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
HEROS.LegacyUserNoCtrlfct	This role determines the rights for remote log-in when user administration is disabled (e.g., via SSH). The control assigns this role automatically.		
	This role grants the rights of the LegacyUser role, with the exception of the following right:		
	■ HEROS.ControlFunctions	■ ctrlfct	■ 337
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the **LegacyUser** role, as well as the following rights:		
	■ HEROS.UserAdmin	■ useradmin	■ 336

NC operator roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Operator	This role allows you to run NC programs.		
	■ NC.OPModeProgramRun	■ NCOpPgmRun	■ 302
NC.Programmer	This role grants the rights of NC programming.		
	This role grants the rights of the Operator role, as well as the following rights:		
	■ NC.EditNCProgram	■ NCEdNCProg	■ 305
	■ NC.EditPalletTable	■ NCEdPal	■ 309
	■ NC.EditPresetTable	■ NCEdPreset	■ 308
	■ NC.EditToolTable	■ NCEdTool	■ 306
	■ NC.OPModeMDi	■ NCOpMDI	■ 301
	■ NC.OPModeManual	■ NCOpManual	■ 300

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Setter	This role allows you to edit the pocket table.		
	This role grants the rights of the Programmer role, as well as the following rights:		
	■ NC.ApproveFsAxis ■ NC.EditPocketTable ■ NC.SetupDrive ■ NC.SetupProgramRun	■ NCApproveFsAxis ■ NCEdPocket ■ NCSetupDrv ■ NCSetupPgRun	■ 319 ■ 307 ■ 315 ■ 303
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the Setter role, as well as the following rights:		
	■ NC.ScheduleProgramRun	■ NCSchedulePgRun	■ 304
NC.LegacyUser	With the **LegacyUser** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **LegacyUser** has the same rights as the AutoProductionSetter.		
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors. ■ Special functions of Q parameter programming and editing the table header Replacement for code number **555343**		
	■ NC.EditNCProgramAdv ■ NC.EditTableAdv	■ NCEditNCPgmAdv ■ NCEditTableAdv	■ 327 ■ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		
	■ NC.RemoteProgramRun	■ NCRemotePgmRun	■ 329

Machine tool builder (PLC) roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
PLC.ConfigureUser	This roles grants the rights on code number **123**.		
	■ NC.ConfigUserAdv ■ NC.SetupDrive	■ NCConfigUserAdv ■ NCSetupDrv	■ 316 ■ 315
PLC.ServiceRead	This role allows read-only access during servicing. This role can be used to display various types of diagnostic information		
	■ NC.Data.AccessServiceRead	■ NCDAServiceRead	■ 324

Refer to your machine manual.

The machine manufacturer can adapt the PLC roles.

When the **Machine tool builder (PLC) roles:** are adapted by the machine manufacturer, the following contents may change:

- The names of the roles
- The number of roles
- The functionality of the roles

43.3.2 List of rights

The table below lists all of the individual rights.

Further information: "Rights", Page 2162

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
HEROS.ReadLogs	Currently no function
NC.OPModeManual	Operating the machine in the **Manual Operation** and **Electronic handwheel** operating modes.
NC.OPModeMDi	Working in the **Positioning w/ Manual Data Input** operating mode.
NC.OpModeProgramRun	Execution of NC programs in the **Program Run Full Sequence** or **Program run, single block** operating mode.
NC.SetupProgramRun	Probing in the **Manual Operation** and **Electronic handwheel** operating modes. Using the **AFC** and **ACC** functions.
NC.ScheduleProgramRun	Programming a scheduled NC program start
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool table
NC.EditPocketTable	Editing the pocket table
NC.EditPresetTable	Editing the preset table
NC.EditPalletTable	Editing pallet tables
NC.SetupDrive	Adjustment of drives by the end user
NC.ApproveFsAxis	Confirming test position of safe axes
NC.EditNCProgramAdv	Additional NC functions
NC.EditTableAdv	Additional table programming functions (e.g., editing of the table head)
HEROS.SetTimezone	Adjustment of date and time, time zone and time synchronization via NTP and the .
HEROS.SetShares	Configuration of public network drives mounted on the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.SetNetwork	Configuration of network and relevant settings for data security
HEROS.BackupUsers	Data backup on the control—for all users configured on the control
HEROS.BackupMachine	Backup and restoring data of the entire machine configuration
HEROS.UserAdmin	Configuration of user administration on the control This includes creating, deleting, and configuring local users

HEROS role name	Description
HEROS.ControlFunctions	Control function of the operating system ■ Auxiliary functions, such as starting and stopping NC software ■ Telemaintenance ■ Advanced diagnostic functions, such as log data
HEROS.SWUpdate	Installation of software updates for the control
HEROS.VMSharedFolders	Access to shared folders of a virtual machine Only relevant when running a programming station within a virtual machine
NC.RemoteProgramRun	NC program start from an external application (e.g., via the DNC interface)
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number **123**
NC.DataAccessServiceRead	Read-only access to the **PLC:** drive during servicing
NC.OpcUaOEMConfiguredDataRead	Read-access through OPC UA NC Server to data defined by the machine manufacturer

43.4 Preassigned error numbers for FN 14: ERROR

With the **FN 14** function you can issue error messages in the NC program.

Further information: "Output error messages with FN 14: ERROR", Page 1381

The following error messages are preassigned by HEIDENHAIN:

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

Error number	Text
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
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

Error number	Text
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as negative
1078	Q303 in meas. cycle undefined!
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory meas. points
1082	Incorrect clearance height
1083	Contradictory plunge type
1084	This fixed cycle not allowed
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not allowed
1090	Enter an infeed not equal to 0
1091	Switchover of Q399 not allowed
1092	Tool not defined
1093	Tool number not permitted
1094	Tool name not permitted
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed
1100	Kinematic access not possible
1101	Meas. pos. not in traverse range
1102	Preset compensation not possible
1103	Tool radius too large
1104	Plunging type is not possible
1105	Plunge angle incorrectly defined
1106	Angular length is undefined
1107	Slot width is too large
1108	Scaling factors not equal
1109	Tool data inconsistent
1110	MOVE not possible
1111	Presetting not allowed!
1112	Thread angle too small!

Error number	Text
1113	3-D 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
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

Error number	Text
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
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

Error number	Text
1189	Chamfer angle cannot be machined with the active tool
1190	The allowances do not define any stock removal
1191	Spindle angle not unique

43.5 System data

43.5.1 List of FN functions

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
		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	-	Label number or label name (string or QS) jumped to in the event of FN14: ERROR with the NC CANCEL reaction instead of aborting the NC program with an error message. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
		3	-	Label number or label name (string or QS) jumped to in the event of an internal server error (SQL, PLC, CFG) or with erroneous file operations (FUNCTION FILECOPY, FUNCTION FILEMOVE, or FUNCTION FILEDELETE) instead of aborting the NC program with an error message. Value = 0: Error has the normal effect.
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	-	Index of prepared tool
		11	-	Index of active tool
		14	-	Number of active spindle
		20	-	Programmed cutting speed in turning operation
		21	-	Spindle mode in turning mode: 0 = constant speed 1 = constant cutting speed
		22	-	Coolant status M7: 0 = inactive, 1 = active
		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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		64	-	Q parameter type for the result (touch probe cycles 30 to 33) 1 = Q, 2 = QL, 3 = QR
		70	-	Multiplier for feed rate (cycles 17 and 18)

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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		28	Tool no.	Maximum speed NMAX
		32	Tool no.	Point angle TANGLE
		34	Tool no.	LIFTOFF allowed (0 = No, 1 = Yes)
		35	Tool no.	Wear tolerance for radius R2TOL
		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
		39	Tool no.	ACC
		40	Tool no.	Pitch for thread cycles
		41	Tool no.	AFC: reference load
		42	Tool no.	AFC: overload early warning
		43	Tool no.	AFC: overload NC stop
		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 lines in a freely definable table that has been opened with FN26: 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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tool radius oversize DR2
		8	-	Tool index
		9	-	Active feed rate
		10	-	Cutting speed [mm/min]
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed with FUNCTION TURNDATA				
	62	1	-	Tool length oversize DXL
		2	-	Tool length oversize DYL
		3	-	Tool length oversize DZL
		4	-	Cutting radius oversize DRS
Values for LAC and VSC				
	71	0	0	Index of the NC axis for which the LAC weighing run will be performed or was last performed (X to W = 1 to 9)
			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
Information about HEIDENHAIN cycles				
	71	20	0	Configuration information for dressing: (**CfgDressSettings**) Maximum search path / set-up clearance
			1	Configuration information for dressing: (**CfgDressSettings**) Search speed (with acoustic emission sensor)
			2	Configuration information for dressing: (**CfgDressSettings**) Feed-rate factor (contact-free motion)
			3	Configuration information for dressing: (**CfgDressSettings**) Feed-rate factor at wheel side
			4	Configuration information for dressing: (**CfgDressSettings**) Feed-rate factor at wheel radius
			5	Tool information for dressing: (**toolgrind.grd**) Set-up clearance in Z (inside)
			6	Tool information for dressing: (**toolgrind.grd**) Set-up clearance in Z (outside)
			7	Machining information for dressing: Set-up clearance in X (diameter)
			8	Machining information for dressing: Ratio of cutting speed
			9	Machining information for dressing: Programmed number of dressing tool
			10	Machining information for dressing: Programmed number of dressing kinematics

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			11	Machining information for dressing: TCPM active/inactive
			12	Machining information for dressing: Programmed position of rotary axis
			13	Machining information for dressing: Cutting speed of the grinding wheel
			14	Machining information for dressing: Rotational speed of dressing spindle
			15	Machining information for dressing: Magazine number of dresser
			16	Machining information for dressing: Pocket number of dresser
		21	0	Configuration information for grinding: (**CfgGrindSettings**) Infeed velocity (synchronous reciprocation)
			1	Configuration information for grinding: (**CfgGrindSettings**) Search speed (with acoustic emission sensor)
			2	Configuration information for grinding: (**CfgGrindSettings**) Relief amount
			3	Configuration information for grinding: (**CfgGrindSettings**) Dimensional control offset
		22	0	Configuration information for behavior when the sensor has not responded. (**CfgGrindEvents/sensorNotReached**) IDX: Sensor
		23	0	Configuration information for behavior when the sensor is already active at the start. (**CfgGrindEvents/sensorActiveAtStart**) IDX: Sensor
		24	1	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource2**) Sensor function = Infeed with touch probe
			2	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource2**) Sensor function = Infeed with acoustic emission sensor
			3	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource2**) Sensor function = Infeed with dimensional control

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			9	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource2**) Sensor function = OEM-specific interaction 1
			10	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource2**) Sensor function = OEM-specific interaction 2
			11	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource2**) Sensor function = Intermediate dressing
			12	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource2**) Sensor function = Teach button
		25	1	Configuration information for the relief amount of a sensor function (**CfgGrindEvents/sensorReleave**) Sensor function = Infeed with touch probe
			2	Configuration information for the relief amount of a sensor function (**CfgGrindEvents/sensorReleave**) Sensor function = Infeed with acoustic emission sensor
			3	Configuration information for the relief amount of a sensor function (**CfgGrindEvents/sensorReleave**) Sensor function = Infeed with dimensional control
			9	Configuration information for the relief amount of a sensor function (**CfgGrindEvents/sensorReleave**) Sensor function = OEM-specific interaction 1
			10	Configuration information for the relief amount of a sensor function (**CfgGrindEvents/sensorReleave**) Sensor function = OEM-specific interaction 2
			11	Configuration information for the relief amount of a sensor function (**CfgGrindEvents/sensorReleave**) Sensor function = Intermediate dressing
			12	Configuration information for the relief amount of a sensor function (**CfgGrindEvents/sensorReleave**) Sensor function = Teach button

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		26	1	Configuration information for the type of reaction to an event of a sensor function (**CfgGrindEvents/sensorReaction**) Sensor function = Infeed with touch probe
			2	Configuration information for the type of reaction to an event of a sensor function (**CfgGrindEvents/sensorReaction**) Sensor function = Infeed with acoustic emission sensor
			3	Configuration information for the type of reaction to an event of a sensor function (**CfgGrindEvents/sensorReaction**) Sensor function = Infeed with dimensional control
			9	Configuration information for the type of reaction to an event of a sensor function (**CfgGrindEvents/sensorReaction**) Sensor function = OEM-specific interaction 1
			10	Configuration information for the type of reaction to an event of a sensor function (**CfgGrindEvents/sensorReaction**) Sensor function = OEM-specific interaction 2
			11	Configuration information for the type of reaction to an event of a sensor function (**CfgGrindEvents/sensorReaction**) Sensor function = Intermediate dressing
			12	Configuration information for the type of reaction to an event of a sensor function (**CfgGrindEvents/sensorReaction**) Sensor function = Teach button
		27	1	Configuration information for the event additionally used by a sensor function (**CfgGrindEvents/sensorSource**) Sensor function = Infeed with touch probe
			2	Configuration information for the event additionally used by a sensor function (**CfgGrindEvents/sensorSource**) Sensor function = Infeed with acoustic emission sensor
			3	Configuration information for the event additionally used by a sensor function (**CfgGrindEvents/sensorSource**) Sensor function = Infeed with dimensional control

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			9	Configuration information for the event additionally used by a sensor function (**CfgGrindEvents/sensorSource**) Sensor function = OEM-specific interaction 1
			10	Configuration information for the event additionally used by a sensor function: (**CfgGrindEvents/sensorSource**) Sensor function = OEM-specific interaction 2
			11	Configuration information for the event additionally used by a sensor function (**CfgGrindEvents/sensorSource**) Sensor function = Intermediate dressing
			12	Configuration information for the event additionally used by a sensor function (**CfgGrindEvents/sensorSource**) Sensor function = Teach button
		28	0	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) Cylindrical grinding: override source for reciprocating movement
			1	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) Cylindrical grinding: override source for infeed movement
			2	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) Surface grinding: override source for reciprocating movement
			3	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) Surface grinding: override source for infeed movement
			4	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) Special grinding: override source for reciprocating movement
			5	Configuration information for the assignment of override sources to grinding functions:

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				(**CfgGrindOverrides**) Special grinding: override source for infeed movement
			6	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) Jig grinding (reciprocating stroke)
			7	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) General movements in the infeed generator (example: general movement with/without sensor)
			8	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) General movements in the infeed generator (example: movement with acoustic emission sensor)
			9	Configuration information for the assignment of override sources to grinding functions: (**CfgGrindOverrides**) General movements in the infeed generator (example: movement with touch probe)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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 and oversize from TOOL CALL	Active length

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
		6	Tool no.	Tool length Index 0= active tool
Coordinate transformations				
	210	1	-	Basic rotation (manual)
		2	-	Programmed rotation
		3	-	Active mirror axis. Bits 0 to 2 and 6 to 8: Axes X, Y, Z and U, V, W
		4	Axis	Active scaling factor Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 - 3 (A, B, C)
		6	-	Tilt working plane in Program Run operating modes 0 = Not active –1 = Active
		7	-	Tilt working plane in Manual operating modes 0 = Not active –1 = Active
		8	QL parameter no.	Angle of misalignment between spindle and tilted coordinate system. Projects the angle specified in the QL parameter from the input coordinate system to the tool coordinate system. If IDX is omitted, the angle 0 is used for projection.
		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 - 3 (rotA, rotB, rotC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read values for OEM offset. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	-	Software limit switch on or off: 0 = on, 1 = off For modulo axes, either both the upper and lower limits or no limit at all must be set.
Read the nominal position in the REF system				
	240	1	Axis	Current nominal position in the REF system
Read the nominal position in the REF system, including offsets (handwheel, etc.)				
	241	1	Axis	Current nominal position in the REF system
Read the current position in the active coordinate system				
	270	1	Axis	Current nominal position in the input system When called while tool radius compensation is active, the function supplies the uncompensated positions for the principal axes X, Y, and Z. If the function is called for a rotary axis and tool radius

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				compensation is active, an error message is issued. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
Read the current position in the active coordinate system, including offsets (handwheel, etc.)				
	271	1	Axis	Current nominal position in the input system
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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	Axis	Angle head: Direction vector of the tool in the basic coordinate system B-CS Index: 1, 2, 3 (X, Y, Z)
		10	Axis	Determine programmable axes. Determine the axis ID associated with the specified axis index (index from CfgAxis/axisList). Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		11	Axis ID	Determine programmable axes. Determine the index of the axis (X = 1, Y = 2, ...) for the specified axis ID Index: Axis ID (index from CfgAxis/axisList)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modify the geometrical behavior				
	310	20	Axis	Diameter programming: –1 = on, 0 = off
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		4	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm:ss
		5	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY
		11	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD
		12	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD
		13	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: hh:mm:ss
		14	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm:ss
		15	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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
			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 GPS setting is active 1 = Any GPS setting is active
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
		1	-	GPS: Basic rotation 0 = Off, 1 = On
		3	Axis	GPS: Mirroring 0 = Off, 1 = On Index: 1 - 6 (X, Y, Z, A, B, C)
		4	-	GPS: Shift in the modified workpiece system 0 = Off, 1 = On
		5	-	GPS: Rotation in input system 0 = Off, 1 = On
		6	-	GPS: Feed rate factor 0 = Off, 1 = On
		8	-	GPS: Handwheel superimpositioning 0 = Off, 1 = On
		10	-	GPS: Virtual tool axis VT 0 = Off, 1 = On
		15	-	GPS: Selection of the handwheel coordinate system 0 = Machine coordinate system M-CS 1 = Workpiece coordinate system W-CS 2 = Modified workpiece coordinate system mW-CS 3 = Working plane coordinate system WPL-CS
		16	-	GPS: Shift in the workpiece system 0 = Off, 1 = On
		17	-	GPS: Axis offset 0 = Off, 1 = On

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 - 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 - 6 (X, Y, Z, A, B, C)
		5	-	GPS: Angle of rotation in input coordinate system I-CS
		6	-	GPS: Feed rate factor
		8	Axis	GPS: Handwheel superimpositioning Maximum value Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		9	Axis	GPS: Value for handwheel superimpositioning Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 - 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 - 6 (A, B, C)
TS touch trigger probe				
	350	50	1	Touch probe type: 0: TS120, 1: TS220, 2: TS440, 3: TS630, 4: TS632, 5: TS640, 6: TS444, 7: TS740
			2	Line in the touch-probe table
		51	-	Effective length
		52	1	Effective radius of the stylus tip
			2	Rounding radius
		53	1	Center offset (reference axis)
			2	Center offset (minor axis)
		54	-	Spindle-orientation angle in degrees (center offset)
		55	1	Rapid traverse
			2	Measuring feed rate
			3	Feed rate for pre-positioning: FMAX_PROBE or FMAX_MACHINE
		56	1	Maximum measuring range
			2	Set-up clearance
		57	1	Spindle orientation possible 0=No, 1=Yes
			2	Angle of spindle orientation in degrees

Group name	Group number ID...	System data number NO....	Index IDX...	Description
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
			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
		6	Coordinate / axis	Readout of the measurement results in the form of coordinates / axis values in the input system from probing opera-tions. Compensation: only length
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
Read values from or write values to the active datum table				
	500	Row number	Column	Read values
Read values from or write values to the preset table (basic transformation)				
	507	Row number	1-6	Read values
Read axis offsets from or write axis offsets to the preset table				
	508	Row number	1-9	Read values
Data for pallet machining				
	510	1	-	Active line
		2	-	Current pallet number. Read value of the NAME column of the last PAL-type entry. If the column is empty or does not contain a numerical value, a value of –1 is returned.
		3	-	Active row of the pallet table.
		4	-	Last line of the NC program for the current pallet.
		5	Axis	Tool-oriented editing: Clearance height is programmed: 0 = No, 1 = Yes Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		6	Axis	Tool-oriented editing: Clearance height The value is invalid if ID510 NR5 returns the value 0 with the corresponding IDX. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		10	-	Row number up to which the pallet table is to be searched during block scan.
		20	-	Type of pallet editing? 0 = Workpiece-oriented 1 = Tool oriented
		21	-	Automatic continuation after NC error: 0 = Locked 1 = Active 10 = Abort continuation 11 = Continuation with the rows in the pallet table that would have been executed next if not for the NC error

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				12 = Continuation with the row in the pallet table in which the NC error arose 13 = Continuation with the next pallet

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data from the point table				
	520	Row number	10	Read value from active point table.
			11	Read value from active point table.
			1-3 X/Y/Z	Read value from active point table.
Read or write the active preset				
	530	1	-	Number of the active preset in the active preset table.
Active pallet preset				
	540	1	-	Number of the active pallet preset. Returns the number of the active preset. If no pallet preset is active, the function returns the value –1.
		2	-	Number of the active pallet preset. As with NR1.
Values for the basic transformation of the pallet preset				
	547	Row number	Axis	Read values of the basic transformation from the pallet preset table. Index: 1 to 6 (X, Y, Z, SPA, SPB, SPC)
Axis offsets from the pallet preset table				
	548	Row number	Offset	Read values of the axis offsets from the pallet preset table. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Read and write the machine status				
	590	2	1-30	Freely available; not deleted during program selection.
		3	1-30	Freely available; not deleted during a power failure (persistent storage).
Read/write look-ahead parameter of a single axis (at machine level)				
	610	1	-	Minimum feed rate (**MP_minPathFeed**) in mm/min
		2	-	Minimum feed rate at corners (**MP_minCornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPathJerk**) in m/s³
		5	-	Max. jerk at high speeds (**MP_maxPathJerkHi**) in m/s³
		6	-	Tolerance at low speeds (**MP_pathTolerance**) in mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tolerance at high speeds (**MP_pathToleranceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPathYank**) in m/s^4
		9	-	Tolerance factor for curve machining (**MP_curveTolFactor**)
		10	-	Factor for max. permissible jerk at curvature changes (**MP_curveJerkFactor**)
		11	-	Maximum jerk with probing movements (**MP_pathMeasJerk**)
		12	-	Angle tolerance for machining feed rate (**MP_angleTolerance**)
		13	-	Angle tolerance for rapid traverse (**MP_angleToleranceHi**)
		14	-	Max. corner angle for polygons (**MP_maxPolyAngle**)
		18	-	Radial acceleration with machining feed rate (**MP_maxTransAcc**)
		19	-	Radial acceleration with rapid traverse (**MP_maxTransAccHi**)
		20	Index of physical axis	Max. feed rate (**MP_maxFeed**) in mm/min
		21	Index of physical axis	Max. acceleration (**MP_maxAcceleration**) in m/s^2
		22	Index of physical axis	Maximum transition jerk of the axis in rapid traverse (**MP_axTransJerkHi**) in m/s^2
		23	Index of physical axis	Maximum transition jerk of the axis during machining free rate (**MP_axTransJerk**) in m/s^3
		24	Index of physical axis	Acceleration feedforward control (**MP_compAcc**)
		25	Index of physical axis	Axis-specific jerk at low speeds (**MP_axPathJerk**) in m/s^3
		26	Index of physical axis	Axis-specific jerk at high speeds (**MP_axPathJerkHi**) in m/s^3
		27	Index of physical axis	More precise tolerance examination in corners (**MP_reduceCornerFeed**) 0 = deactivated, 1 = activated
		28	Index of physical axis	DCM: Maximum tolerance for linear axes in mm (**MP_maxLinearTolerance**)
		29	Index of physical axis	DCM: Maximum angle tolerance in [°] (**MP_maxAngleTolerance**)
		30	Index of physical axis	Tolerance monitoring for successive threads (**MP_threadTolerance**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		31	Index of physical axis	Form (**MP_shape**) of the **axisCutterLoc** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		32	Index of physical axis	Frequency (**MP_frequency**) of the **axisCutterLoc** filter in Hz
		33	Index of physical axis	Form (**MP_shape**) of the **axisPosition** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		34	Index of physical axis	Frequency (**MP_frequency**) of the **axisPosition** filter in Hz
		35	Index of physical axis	Order of the filter for **Manual** operating mode (**MP_manualFilterOrder**)
		36	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisCutterLoc** filter
		37	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisPo-sition** 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 machin-ing 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**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		47	-	Type of acceleration profile (**CfgLaPath/profileType**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal
		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**)

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 effective 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
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC7, TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)
General data of the grinding wheel				
	780	2	-	Width
		3	-	Overhang
		4	-	Alpha angle (optional)
		5	-	Gamma angle (optional)
		6	-	Depth (optional)
		7	-	Rounding radius at the "Further" edge (optional)
		8	-	Rounding radius at the "Nearer" edge (optional)
		9	-	Rounding radius at the "Nearest" edge (optional)
		10	-	Active edge:
		11	-	Type of grinding wheel (straight / angular)
		12	-	External or internal wheel?
		13	-	Compensation angle of the B axis (with respect to the base angle of the location)
		14	-	Type of angular wheel

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		15	-	Total length of the grinding wheel
		16	-	Length of the inner edge of the grinding wheel
		17	-	Minimum wheel diameter (wear limit)
		18	-	Minimum wheel width (wear limit)
		19	-	Tool number
		20	-	Cutting speed
		21	-	Maximum permissible cutting speed
		27	-	Wheel basic type: with relief cut
		28	-	Relief cut on the outside
		29	-	Relief cut on the inside
		30	-	Definition status
		31	-	Radius compensation
		32	-	Compensation of total length
		33	-	Compensation of overhang
		34	-	Compensation for the length to the inner-most edge
		35	-	Radius of the shaft of the grinding wheel
		36	-	Initial dressing performed?
		37	-	Dresser location for initial dressing
		38	-	Dresser tool for initial dressing
		39	-	Has the grinding wheel been measured?
		51	-	Dresser tool for dressing on the diameter
		52	-	Dresser tool for dressing on the outer edge
		53	-	Dresser tool for dressing on the inner edge
		54	-	Dressing of the diameter according to the number of calls
		55	-	Dressing of the outer edge according to the number of calls
		56	-	Dressing of the inner edge according to the number of calls
		57	-	Dressing counter of the diameter
		58	-	Dressing counter of the outer edge
		59	-	Dressing counter of the inner edge
		60	-	Selection of compensation method
		61	-	Inclination angle of dressing tool
		101	-	Radius of grinding wheel

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Datum shift for the grinding wheel				
	781	1	Axis	Datum shift from calibrating the front edges
		2	Axis	Datum shift from calibrating the rear edges
		3	Axis	Datum shift from setup
		4	Axis	Programmed wheel-specific datum shift
		5-9	Axis	Additional wheel-specific datum shift
Geometry of the grinding wheel				
	782	1	-	Wheel shape
		2	-	Overrun on the outer side
		3	-	Overrun on the inner side
		4	-	Overrun diameter
Detailed geometry (contour) of the grinding wheel				
	783	1	1	Chamfer width of the outer side of the wheel
			2	Chamfer width of the inner side of the wheel
		2	1	Chamfer angle of the outer side of the wheel
			2	Chamfer angle of the inner side of the wheel
		3	1	Corner radius of the outer side of the wheel
			2	Corner radius of the inner side of the wheel
		4	1	Side length of the outer side of the wheel
			2	Side length of the inner side of the wheel
		5	1	Relief length of the outer side of the wheel
			2	Relief length of the inner side of the wheel
		6	1	Relief angle of the outer side of the wheel
			2	Relief angle of the inner side of the wheel
		7	1	Recess length of the outer side of the wheel
			2	Recess length of the inner side of the wheel
		8	1	Departing radius of the outer side of the wheel
			2	Departing radius of the inner side of the wheel
		9	1	Total depth on the outside

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			2	Total depth on the inside
Data for dressing the grinding wheel				
	784	1	-	Number of safety positions
		5	-	Dressing method
		6	-	Number of the dressing program
		7	-	Amount of infeed for dressing
		8	-	Angle of infeed / infeed direction for dressing
		9	-	Number of repetitions for dressing
		10	-	Number of idle strokes for dressing
		11	-	Feed rate for dressing on the diameter
		12	-	Feed rate factor for dressing the side (with respect to NR11)
		13	-	Feed rate factor for dressing radii (with respect to NR11)
		14	-	Feed rate factor for dressing angular wheels (with respect to NR11)
		15	-	Feed rate outside the wheel, for pre-profiling
		16	-	Feed rate factor inside the wheel (with respect to NR15), for pre-profiling
		25	-	Dressing method for intermediate dressing
		26	-	Number of the program for intermediate dressing
		27	-	Amount of infeed for intermediate dressing
		28	-	Angle of infeed / infeed direction for intermediate dressing
		29	-	Number of repetitions for intermediate dressing
		30	-	Number of idle strokes for intermediate dressing
		31	-	Feed rate for intermediate dressing

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Safety positions for the grinding wheel				
	785	1	Axis	Safety position no. 1
		2	Axis	Safety position no. 2
		3	Axis	Safety position no. 3
		4	Axis	Safety position no. 4
Data of the dressing tool for the grinding wheel				
	789	1	-	Type
		2	-	Length L1
		3	-	Length L2
		4	-	Radius
		5	-	Orientation: 1=RadType1, 2=RadType2, 3=RadType3
		10	-	Rotational speed of the dressing spindle

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read Functional Safety (FS) information				
	820	1	-	FS limitations: 0 = No Functional Safety (FS) 1 = Guard door open (SOM1) 2 = Guard door open (SOM2) 3 = Guard door open (SOM3) 4 = Guard door open (SOM4) 5 = All guard doors closed
Write data for unbalance monitoring				
	850	10	-	Activate and deactivate unbalance monitoring 0 = unbalance monitoring not active 1 = unbalance monitoring active
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		18	-	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	-	TT: Offset in plane R-OFFS R = 99999.9999
		20	-	TT: Offset in length L-OFFS
		21	-	TT: Break tolerance for length LBREAK
		22	-	TT: Break tolerance for radius RBREAK
		28	-	Maximum spindle speed [rpm] NMAX
		32	-	Point angle TANGLE
		34	-	LIFTOFF allowed (0 = No, 1 = Yes)
		35	-	Wear tolerance for radius R2TOL
		36	-	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	-	Corresponding line in the touch-probe table
		38	-	Timestamp of last use
		39	-	ACC
		40	-	Pitch for thread cycles
		41	-	AFC: reference load
		42	-	AFC: overload early warning
		43	-	AFC: overload NC stop
		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
Read and write data of current turning tool				
	951	1	-	Tool number
		2	-	Tool length XL
		3	-	Tool length YL
		4	-	Tool length ZL
		5	-	Tool length oversize DXL
		6	-	Oversize in tool length DYL
		7	-	Tool length oversize DZL
		8	-	Tooth radius (RS)
		9	-	Tool orientation (TO)
		10	-	Angle of spindle orientation (ORI)
		11	-	Tool angle P_ANGLE
		12	-	Point angle T_ANGLE
		13	-	Recessing width CUT_WIDTH
		14	-	Type (e.g. roughing, finishing, threading, recessing or button tool)
		15	-	Length of cutting edge CUT_LENGTH
		16	-	Compensation of workpiece diameter WPL-DX-DIAM in the working plane coordinate system WPL-CS
		17	-	Compensation of workpiece diameter WPL-DZL in the working plane coordinate system WPL-CS
		18	-	Recessing width oversize
		19	-	Cutting radius oversize
		20	-	Rotation around spatial angle B for offset recessing tools

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Data of the currently active dresser				
	952	1	-	Tool number
		2	-	Tool length XL
		3	-	Tool length YL
		4	-	Tool length ZL
		5	-	Oversize for tool length DXL
		6	-	Oversize for tool length DYL
		7	-	Oversize for tool length DZL
		8	-	Cutter radius
		9	-	Cutting position
		13	-	Cutter width for plate or roll
		14	-	Type (e.g. diamond, plate, spindle, roll)
		19	-	Cutter radius oversize
		20	-	Shaft speed of a dressing spindle or roll

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 NR1
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		19	-	Suppress touch prove movement in cycles: 0 = Movement will be suppressed (CfgMachineSimul/simMode parameter not equal to FullOperation or **Test Run** operating mode is active) 1 = Movement will be performed (CfgMachineSimul/simMode parameter = FullOperation, can be programmed for testing purposes)

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 FN14 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
		21	-	Cancellation during dressing operation for querying within the OEM_CANCEL macro: 0 = Cancellation was not during dressing

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				operation 1 = Cancellation during dressing operation
		30	-	Interpolation of multiple axes permitted? 0 = No (e.g. for straight cut control) 1 = yes
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes
		32	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 does not rotate** error message? **(CfgGeoCycle/displaySpindleErr)** 0 = no, 1 = yes
		2	-	Display the **Check the algebraic sign for depth!** 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. Writing of the value with FN17 concludes Cycle 238. 0 = Circular test 1 = Waterfall diagram 2 = Frequency response 3 = Envelope curve spectrum
			2	Component Monitoring: Index of the axis from **CfgAxes\MP_axisList**
			3 – 9	Component monitoring: further arguments depend on the measurement
		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.
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				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	1 - 16	-	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**)
General data of the grinding wheel				
	10780	1	-	Name of wheel
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 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

43.6 Keycaps for keyboard units and machine operating panels

The keycaps with IDs 12869xx-xx and 1344337-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 361 (FS)

The keycaps with ID 679843-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 360 (FS)

Keycaps for alphabetic keyboard

	ESC	! 1	@ 2	# 3	$ 4	% 5	^ 6	& 7	* 8
ID 1286909	-08	-09	-10	-11	-12	-13	-14	-15	-16

	(9	) 0	_ –	+ =	Q	W	E	R	T
ID 1286909	-17	-18	-19	-20	-21	-22	-23	-24	-25

	Y	U	I	O	P	{ [	}]	¦ \	A
ID 1286909	-26	-27	-28	-29	-30	-31	-32	-33	-34

	S	D	F	G	H	J	K	L	: ;
ID 1286909	-35	-36	–	-38	-39	–	-41	-42	-43
ID 1344337*)	–	–	-01*)	–	–	-02*)	–	–	–

*) With tactile mark

	" '	~ `	Z	X	C	V	B	N	M
ID 1286909	-44	-45	-46	-47	-48	-49	-50	-51	-52

	< ,	> .	? /			ALT	PRT SC	
ID 1286909	-53	-54	-55	-56	-57	-58	-59	-60
ID 679843	–	–	–	-F4	–	–	-F6	–

ID 1286911	-02	-03	-04	-05

ID 1286914	-03

		CTRL
ID 1286915	-02	-03

ID 1286917	-01

Keycaps for operating aids

	PGM MGT		ERR	CALC	MOD	HELP
ID 1286909	-61	-62	-63	-64	-65	-66
ID 679843	–	-36	–	–	–	–

Keycaps for operating modes

ID 1286909	-67	-68	-69	-70	-71	-72	-73	-74
ID 679843	–	–	-66	–	–	–	–	–

Keycaps for programming

	APPR DEP	FK	CHF	L	CR	RND	CT	CC	C
ID 1286909	-75	-76	-77	-78	-79	-80	-81	-82	-83
	TOUCH PROBE	CYCL DEF	CYCL CALL	LBL SET	LBL CALL	STOP	TOOL DEF	TOOL CALL	PGM CALL
ID 1286909	-84	-85	-86	-87	-88	-89	-90	-91	-93
	SPEC FCT								
ID 1286909	-92								
ID 679843	-D6								

Keycaps for axis input and value input

	X	Y	Z	A	B	C	U	V	W
	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange
ID 1286909	-94	-95	-96	-4K	-4Y	-4L	-5K	-98	-4Z
ID 679843	-C8	-D3	-53	-54	-C9	-88	-D4	-31	-55

	IV			ESC	INS		i	X	DEL
	Orange								
ID 1286909	-97	-0N	-3S	-4S	-4T	-3R	-3T	-3U	-3V
ID 679843	-31	-E2	–	–	–	–	–	–	–

	7	8	9	4	5	1	2	3	0
ID 1286909	-0B	-0C	-0D	-0E	–	-0G	-0H	-2L	-2M
ID 1344337*)	–	–	–	–	-03*)	–	–	–	–

*) With tactile mark

	.	−/+	X	Q	CE	DEL	NO ENT	END	
ID 1286909	-0K	-0L	-0M	-2N	-0P	-2P	-0R	-0S	-3N

	>>		P	I
			Orange	
ID 1286909	-3W	-3P	-99	-0A

	ENT
ID 1286914	-04

Keycaps for navigation

		HOME	PG UP		GOTO		END	PG DN
ID 1286909	-0T	-0U	-0V	-0W	–	-0Y	-0Z	-1A
ID 1344337*)	–	–	–	–	-04*)	–	–	–

*) With tactile mark

	↑	→
ID 1344337*)	-06	-07
ID 679843	-42	-41

*) With tactile mark

Keycaps for machine functions

	IV+	Z+	Y+	V+	VI+	X+			Y–
ID 1286909	-1D	-1E	-1F	-1G	-1H	-1K	-1L	-4X	-1N
ID 679843	-09	-07	-05	-11	-13	-03	-16	-E6	-06

	IV–	VI–			FN 1				
ID 1286909	-1P	-1R	-1S	-1T	-1U	-1V	-1W	-1X	-1Y
ID 679843	-10	-14	-23	-22	-24	-29	-02	-21	-20

	FN 2			FN 3			Red	Green	X–
ID 1286909	-1Z	-2A	-2B	-2C	-2D	-2E	-2H	-2K	-2R
ID 679843	-25	-28	-01	-26	-27	-30	-57	-56	-04

		Z–	V–	+	–				
ID 1286909	–	-2T	-2U	-2Z	-3A	-3E	-3F	-3G	-3H
ID 1344337*)	-05*)	–	–	–	–	–	–	–	–
ID 679843	-15	-08	-12	-59	-60	-40	-73	-76	-74

*) With tactile mark

						C+		C–	
ID 1286909	-3L	-3M	-3X	-3Y	-3Z	-4A	-4B	-4C	-4D
ID 679843	-C6	-75	-46	-47	-F2	-67	-51	-68	-99

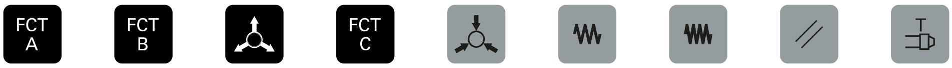
	W+	W–	Red	A+	A–	B+	B–	Red	Red
ID 1286909	-4E	-4F	-4H	-4M	-4N	-4P	-4R	-4U	-06
ID 679843	-B8	-B7	-45	-69	-70	-B2	-B1	-52	-18

	Green	U–	U+			FN 4	FN 5		
ID 1286909	-07	-5A	-5B	-5C	-5D	-4V	-4W	-5E	-5H
ID 679843	-19	-B3	-B4	-61	-62	-A2	-A3	-A4	-E3

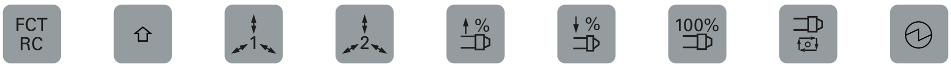
						↑	→	↗
ID 1286909	-5F	-5G	2Y	-3K	-4G	-2V	-2W	-2X
ID 679843	-A5	-A6	–	–	–	–	–	–

ID 679843	-43	-44	-B5	-B6	-B9	-C1	-C2	-C3	-C4

ID 679843	-C5	-D9	-E1	-92	-91	-93	-94	-63	-64



ID 679843	-95	-96	-A1	-C7	-A9	-98	-97	-F3	-72



ID 679843	-E4	-E5	-E7	-E8	-48	-49	-50	-65	-17

	Green	Green	Green	Red	Red
ID 679843	-71	-D8	-90	-89	-D7

	Red	Red
ID 1286909	-2F	-2G

Other keycaps

			Orange	Green	Red				
ID 1286909	-01	-02	-05	-03	-04	–	–	–	–
ID 679843	-33	-34	-35	–	–	-38	-39	-A7	-A8

ID 679843	-D5	-F5

If you need keycaps with additional symbols, please contact HEIDENHAIN.

Index

D

E

F

O

P

U

V

W

HEIDENHAIN

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

Original document
1369999 · 01 · A · 02 · 10/2022 · PDF