HEIDENHAIN

TNC 620

User's Manual for
Cycle Programming

Contents

Contents

Contents

1

Fundamentals

1.1   About this manual

Safety precautions

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

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

⚠ DANGER

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

⚠ WARNING

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

⚠ CAUTION

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

NOTICE

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

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

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

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

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

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

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

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

tnc-userdoc@heidenhain.de

1.2 Control model, software, and features

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

Control model	NC software number
TNC 620	817600-06
TNC 620 E	817601-06
TNC 620 Programming station	817605-06

The suffix E indicates the export version of the control. The export version of the control has the following limitations:

■ Simultaneous linear movement in up to four axes

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

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

■ Tool measurement with the TT

To find out about the actual features of your machine, please contact the machine manufacturer.

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

📖 **Operating instructions:**

All control functions not related to the cycles are described in the TNC 620 User's Manual. This manual is available from HEIDENHAIN upon request.

Conversational Programming User's Manual ID: 1096883-xx

ISO Programming User's Manual ID: 1096887-xx

User's Manual for Setup, Testing and Running NC programs ID: 1263172-xx

---

Software options

The TNC 620 features various software options that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

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

Advanced Function Set 1 (option 8)	
Expanded functions Group 1	**Machining with rotary tables**
	■ Cylindrical contours as if in two axes
	■ Feed rate in distance per minute
	Coordinate conversions:
	Tilting the working plane

Advanced Function Set 2 (option 9)	
Expanded functions Group 2 Export license required	**3-D machining:**
	■ 3-D tool compensation through surface-normal vectors
	■ Using the electronic handwheel to change the angle of the swivel head during program run; the position of the tool point remains unchanged (TCPM = Tool Center Point Management)
	■ Keeping the tool normal to the contour
	■ Tool radius compensation normal to the tool direction
	■ Manual traverse in the active tool-axis system
	Interpolation:
	Linear in > 4 axes (export license required)

Touch Probe Functions (option 17)	
Touch probe functions	**Touch probe cycles:**
	■ Compensation of tool misalignment in automatic mode
	■ Presetting in the **Manual operation** mode
	■ Presetting in automatic mode
	■ Automatically measuring workpieces
	■ Tools can be measured automatically

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

Advanced Programming Features (option 19)	
Expanded programming functions	**FK free contour programming:**
	Programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC

Advanced Programming Features (option 19)

	Fixed cycles:
	■ Peck drilling, reaming, boring, counterboring, centering (cycles 201 to 205, 208, 240, 241)
	■ Milling of internal and external threads (cycles 262 to 265, 267)
	■ Finishing of rectangular and circular pockets and studs (cycles 212 to 215, 251 to 257)
	■ Clearing level and oblique surfaces (cycles 230 to 233)
	■ Straight slots and circular slots (cycles 210, 211, 253, 254)
	■ Linear and circular point patterns (cycles 220, 221)
	■ Contour train, contour pocket—also with contour-parallel machining, trochoidal slot (cycles 20 to 25, 275)
	■ Engraving (cycle 225)
	■ OEM cycles (special cycles developed by the machine tool builder) can be integrated

---

Advanced Graphic Features (option 20)

Expanded graphic functions	**Program-verification graphics, program-run graphics**
	■ Plan view
	■ Projection in three planes
	■ 3-D view

Advanced Function Set 3 (option 21)

Expanded functions Group 3	**Tool compensation:**
	M120: Radius-compensated contour look ahead for up to 99 NC blocks (LOOK AHEAD)
	3-D machining:
	M118: Superimpose handwheel positioning during program run

Pallet Management (option 22)

Pallet management	Processing workpieces in any sequence

Display Step (option 23)

Display step	**Input resolution:**
	■ Linear axes down to 0.01 µm
	■ Rotary axes to 0.00001°

CAD Import (option 42)

CAD import	■ Support for DXF, STEP and IGES
	■ Adoption of contours and point patterns
	■ Simple and convenient specification of presets
	■ Selecting graphical features of contour sections from conversational programs

---

KinematicsOpt (option 48)

Optimizing the machine kinematics	■ Backup/restore active kinematics
	■ Test active kinematics
	■ Optimize active kinematics

Extended Tool Management (option 93)

Extended tool management	Python-based

Remote Desktop Manager (option 133)

Remote operation of external computer units	▪ Windows on a separate computer unit ▪ Incorporated in the control's interface

State Reporting Interface – SRI (option 137)

HTTP accesses to the control status	▪ Reading out the times of status changes ▪ Reading out the active NC programs

Cross Talk Compensation – CTC (option 141)

Compensation of axis couplings	▪ Determination of dynamically caused position deviation through axis acceleration ▪ Compensation of the TCP (**T**ool **C**enter **P**oint)

Position Adaptive Control – PAC (option 142)

Adaptive position control	▪ Changing of the control parameters depending on the position of the axes in the working space ▪ Changing of the control parameters depending on the speed or acceleration of an axis

Load Adaptive Control – LAC (option 143)

Adaptive load control	▪ Automatic determination of workpiece weight and frictional forces ▪ Changing of control parameters depending on the actual mass of the workpiece

Active Chatter Control – ACC (option 145)

Active chatter control	Fully automatic function for chatter control during machining

Active Vibration Damping – AVD (option 46)

Active vibration damping	Damping of machine oscillations to improve the workpiece surface

Batch Process Manager (option 154)

Batch process manager	Planning of production orders

Component Monitoring (option 155)

Component monitoring without external sensors	Monitoring configured machine components for overload

Feature Content Level (upgrade functions)

Along with software options, significant further improvements of the control software are managed via the Feature Content Level **(FCL)** upgrade functions. Functions subject to the FCL are not available simply by updating the software on your control.

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

Upgrade functions are identified in the manual with **FCL n,** where **n** indicates the sequential number of the feature content level.

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

Intended place of operation

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

Legal information

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

▶ Programming operating mode

▶ MOD function

▶ **License Info** softkey

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, they are always provided at the end of the cycle definition. The section "New or changed cycle functions of software 81760x-06 " 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 after a software update, you can add optional Q parameters in cycles where needed. The following steps describe how this is done.

To insert optional Q parameters in existing programs:

- Call the cycle definition
- Press the right cursor key until the new Q parameters are displayed
- Apply the default value or enter a value
- To transfer the new Q parameter, exit the menu by pressing the right arrow key once again or by pressing END
- If you do not wish to apply the new Q parameter, press the NO ENT key

Compatibility

Most NC programs created with older HEIDENHAIN contouring controls (as of TNC 150 B) can be run with the new software version of the TNC 620. Even if new optional parameters ("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 round, 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 be downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

New and changed cycle functions of software 81760x-05

- New Cycle 441 FAST PROBING. With this cycle you can set various touch probe parameters (e.g. positioning feed rate) that are globally effective for all subsequently used touch probe cycles. see "FAST PROBING (Cycle 441, ISO G441, software option 17)", Page 526
- New Cycle 276 3-D Contour Train see "THREE-D CONT. TRAIN (Cycle 276, ISO: G276, software option 19)", Page 245
- Enhancement of the contour train: Cycle 25 with machining of residual material, the cycle was expanded with the following parameters: Q18, Q446, Q447, Q448 see "CONTOUR TRAIN (Cycle 25, ISO: G125, software option 19)", Page 241
- Cycles 256 RECTANGULAR STUDand 257 CIRCULAR STUD were extended by the Q215, Q385, Q369, and Q386 parameters. see "RECTANGULAR STUD (Cycle 256, ISO: G256, software option 19)", Page 181, see "CIRCULAR STUD (Cycle 257, ISO: G257, software option 19)", Page 186
- Cycle 239 can ascertain the current load of the machine axes with the control function LAC. Cycle 239 can now also modify the maximum axis acceleration. Cycle 239 supports the determination of the load on synchronized axes. see "ASCERTAIN THE LOAD (Cycle 239, ISO: G239, software option 143)", Page 343
- Cycles 205 and 241: The feed rate behavior was modified. see "SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, ISO: G241, software option 19)", Page 103, see "UNIVERSAL PECKING (Cycle 205, ISO: G205, software option 19)", Page 92
- Detail changes with Cycle 233: Monitors length of the cutting edge (LCUTS) with finishing operations, when roughing with the milling strategy between 0 to 3, the surface in the milling direction is increased by the value from Q357 (provided that no limitation is set in this direction) see "FACE MILLING (Cycle 233, ISO: G233, software option 19)", Page 196
- CONTOUR DEF can be programmed in ISO format
- The technically obsolete cycles 1, 2, 3, 4, 5, 17, 212, 213, 214, 215, 210, 211, 230, 231 subordinated in "old cycles" can no longer be inserted via the editor. These cycles can however still be modified and executed.
- The tool touch probe cycles 480, 481, 482, 483, 484 can be hidden, see "Setting machine parameters", Page 566
- Cycle 225 Engraving can engrave the current counter reading with a new syntax see "Engraving the counter reading", Page 337
- New column SERIAL in the touch-probe table see "Touch probe data", Page 357

New or changed cycle functions of software 81760x-06

- New Cycle 1410 PROBING ON EDGE (software option 17), see "PROBING ON EDGE (Cycle 1410, ISO: G1410, software option 17)", Page 370

- New Cycle 1411 PROBING TWO CIRCLES (software option 17),see "PROBING TWO CIRCLES (Cycle 1411, ISO G1411, software option 17)", Page 374

- New Cycle 1420 PROBING IN PLANE (software option 17), see "PROBING IN PLANE (Cycle 1420, ISO: G1420, software option 17)", Page 366

- In Cycle 24 SIDE FINISHING, a tangential helix will be used for rounding in the last infeed step, see "SIDE FINISHING (Cycle 24, ISO: G124, software option 19)", Page 238

- Cycle 233 FACE MILLING, was extended by parameter Q367, SURFACE POSITION, see "FACE MILLING (Cycle 233, ISO: G233, software option 19)", Page 196

- Cycle 257 CIRCULAR STUD, now uses Q207 FEED RATE FOR MILLNG for roughing, too, see "CIRCULAR STUD (Cycle 257, ISO: G257, software option 19)", Page 186

- The touch probe cycles 408 to 419 consider chkTiltingAxes (no. 204600) for presetting, see "Touch Probe Cycles: Automatic Presetting", Page 405

- Touch probe cycles 41x, automatic presetting: new behavior of cycle parameters Q303 MEAS. VALUE TRANSFER and Q305 NUMBER IN TABLE, see "Touch Probe Cycles: Automatic Presetting", Page 405

- When pre-positioning the tool with Cycle 420 MEASURE ANGLE, the data both from the cycle and from the touch probe table will be taken into account, see "MEASURE ANGLE (Cycle 420, ISO: G420, software option 17)", Page 472

- Cycle 450 SAVE KINEMATICS, no longer writes identical values when restoring, see "SAVE KINEMATICS (Cycle 450, ISO: G450, option)", Page 534

- In Cycle 451 MEASURE KINEMATICS, the value 3 was added to cycle parameter Q406 MODE, see "MEASURE KINEMATICS (Cycle 451, ISO: G451, option)", Page 537

- In Cycles 451 MEASURE KINEMATICS , the radius of the calibration sphere is monitored in the second measurement only, see "MEASURE KINEMATICS (Cycle 451, ISO: G451, option)", Page 537

- A REACTION column was added to the touch probe table, see "Touch-probe table", Page 356

- The CfgThreadSpindle machine parameter (no. 113600) is now available, see "TAPPING with a floating tap holder (Cycle 206, ISO: G206)", Page 117 , see "TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)", Page 120, see "TAPPING WITH CHIP BREAKING (Cycle 209, ISO: G209, software option 19)", Page 125 , see "THREAD CUTTING (Cycle 18, ISO: G18 , software option 19)", Page 345

2

**Fundamentals /
Overviews**

2.1 Introduction

Frequently recurring machining cycles that comprise several working steps are stored in the control's memory as standard cycles. Coordinate transformations and several special functions are also available as cycles. Most cycles use Q parameters as transfer parameters.

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

▶ You should run a program test before machining

---

i If you use indirect parameter assignments in cycles with numbers greater than 200 (e.g. **Q210 = Q1**), any change in the assigned parameter, e.g. in **Q1**) will have no effect after the cycle definition. Define the cycle parameter (e.g. **Q210**) directly in such cases.

If you define a feed-rate parameter for machining cycles with numbers greater than 200, then instead of entering a numerical value, you can use soft keys to assign the feed rate defined in the **TOOL CALL** block (**FAUTO** soft key). You can also use the feed-rate alternatives **FMAX** (rapid traverse), **FZ** (feed per tooth), and **FU** (feed per rev), depending on the respective cycle and the function of the feed-rate parameter.

Note that, after a cycle definition, a change of the **FAUTO** feed rate has no effect, because internally, the control assigns the feed rate from the **TOOL CALL** block when processing the cycle definition.

If you want to delete a cycle that includes multiple sub-blocks, the control prompts you whether you want to delete the whole cycle.

2.2 Available cycle groups

Overview of fixed cycles

| CYCL DEF | ▶ The soft-key row shows the available groups of cycles |

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring and counterboring	72
DRILLING/ THREAD	Cycles for tapping, thread cutting and thread milling	116
POCKETS/ STUDS/ SLOTS	Cycles for milling pockets, studs and slots and for face milling	156
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	298
SL CYCLES	SL (Subcontour List) cycles for machining contours that consist of multiple overlapping subcontours as well as cycles for cylinder surface machining and trochoidal milling	266
PATTERN	Cycles for producing point patterns, such as circular or linear hole patterns	210
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, determining the load,	324

| ▷ | ▶ If required, switch to machine-specific fixed cycles. These fixed cycles can be integrated by your machine tool builder. |

Overview of touch probe cycles

TOUCH PROBE

▶ The soft-key row shows the available groups of cycles

Soft key	Cycle group	Page
ROTATION	Cycles for automatic measurement and compensation of workpiece misalignment	359
PRESET	Cycles for automatic workpiece presetting	406
MEASURING	Cycles for automatic workpiece inspection	464
SPECIAL CYCLES	Special cycles	508
CALIBRATE TS	Touch probe calibration	515
KINEMATICS	Cycles for automatic kinematics measurement	529
TT CYCLES	Cycles for automatic tool measurement (enabled by the machine manufacturer)	564

▶ If required, switch to machine-specific touch probe cycles. These touch probe cycles can be integrated by your machine tool builder.

3

Using Fixed Cycles

3.1 Working with fixed cycles

Machine-specific cycles (software option19)

Cycles are available at many machines. Your machine manufacturer implements these cycles into the control, in addition to the HEIDENHAIN cycles. These cycles are available in a separate cycle-number range:

- Cycles 300 to 399
  Machine-specific cycles that are to be defined through the **CYCLE DEF** key
- Cycles 500 to 599
  Machine-specific touch probe cycles that are to be defined through the **TOUCH PROBE** key

> Refer to your machine manual for a description of the specific functionality.

Some machine-specific cycles use transfer parameters that are also part of HEIDENHAIN standard cycles. In order to avoid problems (related to overwriting of transfer parameters that are used more than once), when using DEF-active cycles (cycles that the control runs automatically during cycle definition) and CALL-active cycles (cycles that you need to call to run them) used at the same time,

Further information: "Calling a cycle", Page 52

do the following: The following procedure is advisable:

- ▶ As a rule, always program DEF-active cycles before CALL-active cycles
- ▶ Only program a DEF-active cycle between the definition of a CALL-active cycle and the cycle call if there are no interferences of transfer parameters of these two cycles

Defining a cycle using soft keys

CYCL DEF

▶ The soft-key row shows the available groups of cycles

DRILLING/ THREAD

▶ Select the desired cycle group, e.g. drilling cycles

262

▶ Select the cycle, e.g. **THREAD MILLING**. The control initiates a dialog and prompts you for all required input values. At the same time, a graphic is displayed in the right half of the screen. The required parameter is highlighted

▶ Enter all parameters required by the control. Conclude each input with the **ENT** key

▶ The control closes the dialog when all required data has been entered
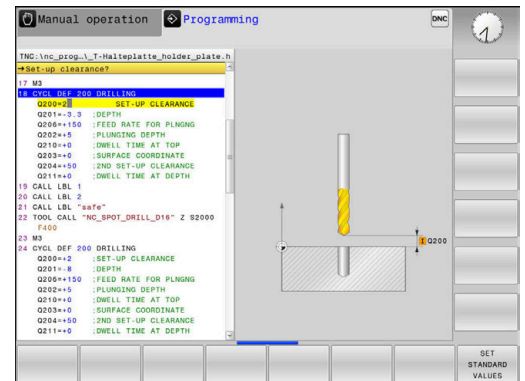
Defining a cycle using the GOTO function

CYCL DEF

▶ The soft-key row shows the available groups of cycles

GOTO

▶ The control displays a pop-up window with an overview of the cycles

▶ Select the desired cycle with the cursor keys or

▶ Enter the cycle number. Confirm each input with the **ENT** key. The control then initiates the cycle dialog as described above

Example

7 CYCL DEF 200 DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=3	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q210=0	;DWELL TIME AT TOP
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q211=0.25	;DWELL TIME AT DEPTH
Q395=0	;DEPTH REFERENCE

Calling a cycle

	Requirements
i	Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Direction of rotation of the spindle (M3/M4)
- Cycle definition (CYCL DEF)

For some cycles, additional prerequisites must be observed. They are detailed in the descriptions for each cycle.

The following cycles become effective automatically as soon as they have been defined in the program. You cannot and must not call them:

- Cycle 220 for point patterns on circles and Cycle 221 for point patterns on lines
- SL Cycle 14 CONTOUR
- SL Cycle 20 CONTOUR DATA
- Cycle 32 TOLERANCE
- Cycles for coordinate transformation
- Cycle 9 DWELL TIME
- All touch probe cycles

You can call all other cycles with the functions described as follows.

Calling a cycle with CYCL CALL

The **CYCL CALL** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that was programmed last before the CYCL CALL block.

CYCL CALL	▶ To program the cycle call: Press the **CYCL CALL** key
	▶ To enter the cycle call: Press the **CYCL CALL M** soft key
	▶ If necessary, enter the M function (miscellaneous function) (such as **M3** to switch the spindle on), or close the dialog by pressing the **END** key

Calling a cycle with CYCL CALL PAT

The **CYCL CALL PAT** function calls the most recently defined machining cycle at all positions that you defined in a PATTERN DEF pattern definition or in a point table.

Further information: "Pattern definition with PATTERN DEF", Page 59

Further information: "Point tables", Page 66

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.

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

i 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 fixed cycle.

If the control is to execute the cycle automatically after every positioning block, program the first cycle call with **M89**.

To cancel the effect of **M89**, program the following:

- **M99** in the positioning block in which you move to the last starting point, or
- Use **CYCL DEF** to define a new fixed cycle

i In combination with FK programming, the control does not support M89!

Working with a parallel axis

The control performs infeed movements in the parallel axis (W axis) that was defined in the **TOOL CALL** block as the spindle axis. The status display shows "W", the tool calculation is performed in the W axis.

This is only possible when programming the following cycles:

Cycle	W axis function
200 **DRILLING**	■
201 **REAMING**	■
202 **BORING**	■
203 **UNIVERSAL DRILLING**	■
204 **BACK BORING**	■
205 **UNIVERSAL PECKING**	■
208 **BORE MILLING**	■
225 **ENGRAVING**	■
232 **FACE MILLING**	■
233 **FACE MILLING**	■
241 **SINGLE-LIP D.H.DRLNG**	■

> **ℹ** HEIDENHAIN recommends not to use **TOOL CALL W**! Use **FUNCTION PARAXMODE** or **FUNCTION PARAXCOMP**.
>
> **Further information:** User's Manual for Conversational Programming

3.2 Program defaults for cycles

Overview

All Cycles 20 to 25, as well as all of those with numbers 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** you can define these cycle parameters at the beginning of the program, so that they are effective globally for all machining cycles used in the NC program. In the respective machining cycle, you then simply reference the value defined at the beginning of the program.

The following GLOBAL DEF functions are available:

Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	57
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	57
110 GLOBAL DEF POCKT MLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	57
111 GLOBAL DEF CNTR MLLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	57
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	58
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	58

Entering GLOBAL DEF

	▶ Operating mode: Press the **Programming** key
SPEC FCT	▶ Press the **SPEC FCT** key to select the special functions
PROGRAM DEFAULTS	▶ Select the functions for program defaults
GLOBAL DEF	▶ Press the **GLOBAL DEF** soft key
100 GLOBAL DEF GENERAL	▶ Select the desired GLOBAL DEF function, e.g. by pressing the **GLOBAL DEF GENERAL** soft key
	▶ Enter the required definitions, and confirm each entry with the **ENT** key
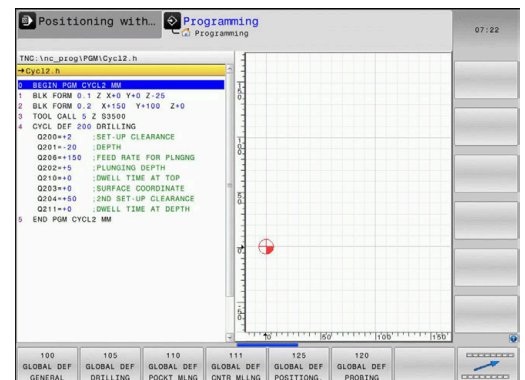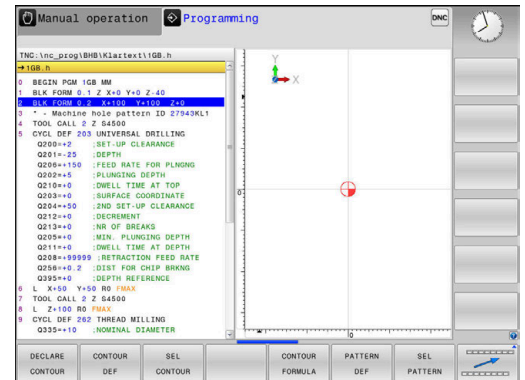
Using GLOBAL DEF information

If you entered the respective GLOBAL DEF functions at the start of the program, you can reference these globally valid values when defining any machining cycle.

Proceed as follows:

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

▶ Select machining cycles: Press the **CYCLE DEF** key

▶ Select the desired cycle group, e.g. drilling cycles

▶ Select the desired cycle, e.g. **drilling**

▶ If a global parameter exists, the control will display the **SET STANDARD VALUES** soft key

▶ Press the **SET STANDARD VALUES** soft key. The control enters the word **PREDEF** (predefined) in the cycle definition. This creates a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program

---
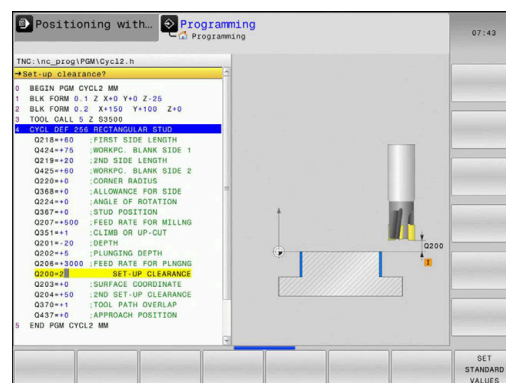
NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly.

▶ Make sure to use **GLOBAL DEF** carefully. Test your program before executing it

▶ If a fixed value is entered in machining cycles, **GLOBAL DEF** does not modify this value

---

Global data valid everywhere

▶ **Safety clearance**: Distance between tool face and workpiece surface for automated approach of the cycle start position in the tool axis

▶ **2nd set-up clearance:** Position to which the control positions the tool at the end of a machining step. The next machining position is approached at this height in the working plane

▶ **F positioning:** Feed rate at which the control traverses the tool within a cycle

▶ **F retraction:** Feed rate at which the control retracts the tool

ⓘ The parameters are valid for all fixed cycles with numbers greater than 2xx.

Global data for drilling operations

▶ **Retraction rate for chip breaking:** Value by which the control retracts the tool during chip breaking

▶ **Dwell time at depth**: Time in seconds that the tool remains at the hole bottom

▶ **Dwell time at top**: Time in seconds that the tool remains at the set-up clearance.

ⓘ The parameters apply to the drilling, tapping and thread milling cycles 200 to 209, 240, 241 and 262 to 267.

Global data for milling operations with pocket cycles 25x

▶ **Overlap factor**: The tool radius multiplied by the overlap factor equals the stepover

▶ **Climb or up-cut**: Select the type of milling

▶ **Plunging type**: Plunge into the material helically, in a reciprocating motion, or vertically

ⓘ The parameters apply to milling cycles 251 to 257.

Global data for milling operations with contour cycles

▶ **Set-up clearance**: Distance between tool face and workpiece surface for automated approach of the cycle start position in the Tool axis

▶ **Clearance height**: Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle)

▶ **Overlap factor**: The tool radius multiplied by the overlap factor equals the stepover

▶ **Climb or up-cut**: Select the type of milling

ⓘ The parameters apply to SL cycles 20, 22, 23, 24 and 25.

Global data for positioning behavior

▶ **Positioning behavior**: Retraction in the tool axis at the end of a machining step, return to the 2nd set-up clearance or to the position at the beginning of the unit

> 🛈 The parameters apply to each fixed cycle that you call with the **CYCL CALL PAT** function.

Global data for probing functions

▶ **Set-up clearance**: Distance between stylus and workpiece surface for automated approach of the probing position

▶ **Clearance height:** The coordinate in the touch probe axis to which the control traverses the touch probe between measuring points if the **Move to clearance height** option is activated

▶ **Move to clearance height**: Select whether the control moves the touch probe to the set-up clearance or clearance height between the measuring points

> 🛈 The parameters apply to all touch probe cycles numbered 4xx.

3.3 Pattern definition with PATTERN DEF

Application

You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. Just like in cycle definitions, help graphics are available for pattern definition that clearly indicate the input parameters required.

> **NOTICE**
>
> **Danger of collision!**
>
> The **PATTERN DEF** function calculates the machining coordinates in the **X** and **Y** axes For all tools axes apart from **Z** there is a danger of collision in the following operation!
>
> ▶ Use **PATTERN DEF** only in connection with the tool axis **Z**

The following machining patterns are available:

Soft key	Machining pattern	Page
POINT	POINT Definition of up to any 9 machining positions	61
ROW	ROW Definition of a single row, straight or rotated	61
PATTERN	PATTERN Definition of a single pattern, straight, rotated or distorted	62
FRAME	FRAME Definition of a single frame, straight, rotated or distorted	63
CIRCLE	CIRCLE Definition of a full circle	64
PITCH CIR	PITCH CIRCLE Definition of a pitch circle	65

Entering PATTERN DEF

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

▶ Press the **SPEC FCT** key to select the special functions
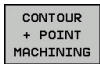
▶ Select the functions for contour and point machining

▶ Press the **PATTERN DEF** soft key
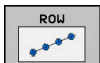
▶ Select the desired machining pattern, e.g. press the "single row" soft key

▶ Enter the required definitions, and confirm each entry with the **ENT** key

Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function.

Further information: "Calling a cycle", Page 52

The control then performs the most recently defined machining cycle based on the machining pattern you defined.

> ℹ️ A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.
>
> You can use the mid-program startup function to select any point at which you want to start or continue machining.
> **Further information**: User's Manual for Setup, Testing and Running NC programs
>
> The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.
>
> If the coordinate surface in PATTERN DEF is larger than in the cycle, the 2nd set-up clearance references the coordinate surface in PATTERN DEF.
>
> If the coordinate surface in the cycle is larger than in PATTERN DEF, the set-up clearance references the sum of both coordinate surfaces.
>
> Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with Q352=1 (found under **SPEC FCT**/ Program Parameters). If you do so, the control will always position the tool at the second set-up clearance defined in the cycle.

Defining individual machining positions

ℹ You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.

POS1 must be programmed with absolute coordinates. POS2 to POS9 can be programmed as absolute and/or incremental values.

If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

```
POINT
 ◆
```
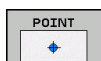
▶ POS1: **X coord. of machining position** (absolute): Enter the X coordinate

▶ POS1: **Y coord. of machining position** (absolute): Enter the Y coordinate

▶ POS1: **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

▶ POS2: **X coord. of machining position** (absolute or incremental): Enter the X coordinate

▶ POS2: **Y coord. of machining position** (absolute or incremental): Enter the Y coordinate

▶ POS2: **Coordinate of workpiece surface** (absolute or incremental): Enter the Z coordinate

Example

10 L Z+100 R0 FMAX

11 PATTERN DEF
POS1 (X+25 Y+33.5 Z+0)
POS2 (X+15 IY+6.5 Z+0)
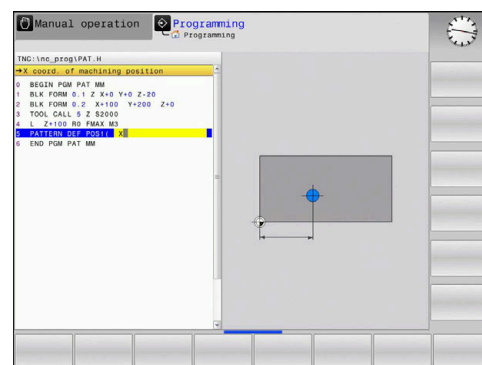
Defining a single row

ℹ 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.

```
ROW
•••
```
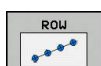
▶ **Starting point in X** (absolute): Coordinate of the pattern row starting point in the X axis

▶ **Starting point in Y** (absolute): Coordinate of the pattern row starting point in the Y axis

▶ **Spacing of machining positions** (incremental): Distance between the machining positions. You can enter a positive or negative value

▶ **Number of operations**: Total number of machining positions

▶ **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value

▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Example

10 L Z+100 R0 FMAX

11 PATTERN DEF ROW1
(X+25 Y+33.5 D+8 NUM5 ROT+0 Z +0)
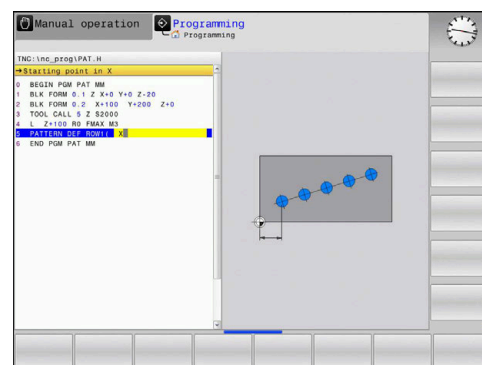
Defining a single pattern

If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.
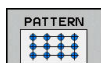
- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- ▶ **Spacing of machining positions X** (incremental): Distance between the machining positions in the X direction. You can enter a positive or negative value
- ▶ **Spacing of machining positions Y** (incremental): Distance between the machining positions in the Y direction. You can enter a positive or negative value
- ▶ **Number of columns**: Total number of columns in the pattern
- ▶ **Number of rows**: Total number of rows in the pattern
- ▶ **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
- ▶ **Rotary pos. ref. ax.**: Angle of rotation about which only the reference axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- ▶ **Rotary pos. minor ax.**: Angle of rotation about which only the minor axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value.
- ▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Example

10 L Z+100 R0 FMAX

11 PATTERN DEF PAT1 (X+25 Y+33,5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0)
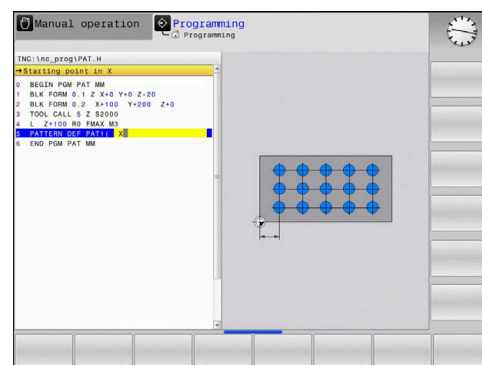
Defining individual frames

ℹ If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.

▶ **Starting point in X** (absolute): Coordinate of the frame starting point in the X axis

▶ **Starting point in Y** (absolute): Coordinate of the frame starting point in the Y axis

▶ **Spacing of machining positions X** (incremental): Distance between the machining positions in the X direction. You can enter a positive or negative value

▶ **Spacing of machining positions Y** (incremental): Distance between the machining positions in the Y direction. You can enter a positive or negative value

▶ **Number of columns**: Total number of columns in the pattern

▶ **Number of rows**: Total number of rows in the pattern

▶ **Rot. position of entire pattern** (absolute): Angle of rotation by which the entire pattern is rotated about the entered starting point. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value

▶ **Rotary pos. ref. ax.**: Angle of rotation about which only the reference axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value.

▶ **Rotary pos. minor ax.**: Angle of rotation about which only the minor axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value.

▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Defining a full circle

ℹ️ If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
▶ **Bolt-hole circle diameter**: Diameter of the circular hole pattern
▶ **Starting angle**: Polar angle of the first machining position. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
▶ **Number of operations**: Total number of machining positions on the circle
▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Example

10 L Z+100 R0 FMAX
11 PATTERN DEF CIRC1   (X+25 Y+33 D80 START+45 NUM8 Z +0)
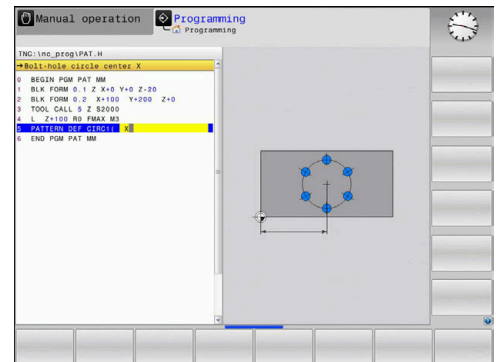
Defining a pitch circle

ℹ️ 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.

PITCH CIR

▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
▶ **Bolt-hole circle diameter**: Diameter of the circular hole pattern
▶ **Starting angle**: Polar angle of the first machining position. Reference axis: Principal axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value
▶ **Stepping angle/Stopping angle**: Incremental polar angle between two machining positions. You can enter a positive or negative value. As an alternative, you can enter the end angle (switch via soft key)
▶ **Number of operations**: Total number of machining positions on the circle
▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Example

10 L Z+100 R0 FMAX
11 PATTERN DEF PITCHCIRC1 (X+25 Y+33 D80 START+45 STEP30 NUM8 Z+0)
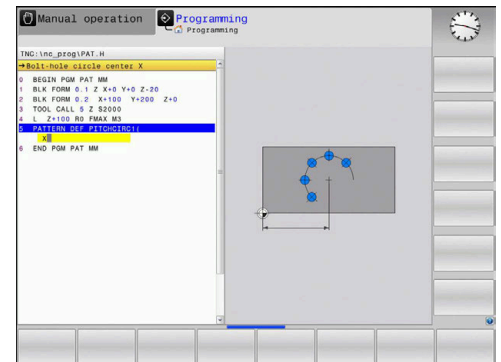
3.4 Point tables

Application

You should create a point table whenever you want to run a cycle or several cycles in sequence, to machine an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting point coordinates of the respective cycle (e.g. center coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Entering values into a point table

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

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

FILE NAME?

▶ Enter the name and file type of the point table. Confirm with the **ENT** key

▶ Select the unit of measure: Press the **MM** or **INCH** soft key. The control changes to the program window and displays an empty points table
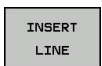
▶ Press the **INSERT LINE** soft key to insert a new line. Enter the coordinates of the desired machining position

Repeat the process until all desired coordinates have been entered.

> ⓘ   The name of the point table must begin with a letter.
>
> Use the **SORT/ HIDE COLUMNS** soft keys (fourth soft-key row) to specify which coordinates you want to enter into the point table.

Hiding single points from the machining process

In the **FADE** column of the point table you can specify if the defined point is to be hidden during the machining process.

↓	▶	In the table, select the point to be hidden
↑		
→	▶	Select the **FADE** column
ENT	▶	Activate hiding or
NO ENT	▶	Deactivate hiding

Selecting a point table in the NC program

In the **Programming** mode of operation, select the NC program for which you want to activate the point table:

PGM CALL	▶	Press the **PGM CALL** key to call the function for selecting the point table
SELECT POINT TABLE	▶	Press the **SELECT POINT TABLE** soft key
SELECT FILE	▶	Press the **SELECT FILE** soft key
	▶	Select the point table and confirm with the **OK** soft key
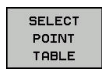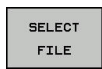
If the point table is not stored in the same directory as the NC program, you must enter the complete path.

Example

```
7 SEL PATTERN  "TNC:\DIRKT5\NUST35.PNT"
```

Calling a cycle in connection with point tables

If you want the control to call the cycle at the points that you last defined in a point table, then program the cycle call with **CYCLE CALL PAT**:

CYCL CALL

▶ To program the cycle call: Press the **CYCL CALL** key

▶ To call the point table, press the **CYCL CALL PAT** soft key

▶ Enter the feed rate at which the control is to move from point to point or press the **F MAX** soft key (if you make no entry, the control will move at the last programmed feed rate)

▶ Enter a miscellaneous function (M function) if required. Confirm your input with the **END** key

The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.

Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with Q352=1 (found under **SPEC FCT**/Program Parameters). If you do so, the control will always position the tool at the second set-up clearance defined in the cycle.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the M103 miscellaneous function.

Effect of the point table with SL cycles and Cycle 12

The control interprets the points as an additional datum shift.

Effect of the point table with Cycles 200 to 208, and 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 spindle axis, you must define the coordinate of the workpiece upper edge (Q203) as 0.

Effect of the point table with 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 spindle axis, you must define the coordinate of the workpiece upper edge (Q203) as 0.

> **i** If you call **CYCL CALL PAT**, the control will use the point table that you defined last. This is also the case if you defined the point table in an NC program nested with **CALL PGM**.

NOTICE

Danger of collision!

If you program a clearance height for any points in a point table, the control will ignore the 2nd set-up clearance for **all** points of this machining cycle!

▶ Program GLOBAL DEF 125 POSITIONING beforehand. This will ensure that the control considers the clearance height from the point table for the corresponding point only.

4

Fixed Cycles: Drilling

4.1 Fundamentals

Overview

The control provides the following cycles for all types of drilling operations:

Soft key	Cycle	Page
	240 CENTERING With automatic pre-position-ing, 2nd set-up clearance, optional entry of the centering diameter or centering depth	73
	200 DRILLING With automatic pre-position-ing, 2nd set-up clearance	75
	201 REAMING With automatic pre-position-ing, 2nd set-up clearance	77
	202 BORING With automatic pre-position-ing, 2nd set-up clearance	79
	203 UNIVERSAL DRILLING With automatic pre-position-ing, 2nd set-up clearance, chip breaking, and decrementing	82
	204 BACK BORING With automatic pre-position-ing, 2nd set-up clearance	88
	205 UNIVERSAL PECKING With automatic pre-position-ing, 2nd set-up clearance, chip breaking, and advanced stop distance	92
	208 BORE MILLING With automatic pre-position-ing, 2nd set-up clearance	100
	241 SINGLE-LIP D.H.DRLNG With automatic pre-position-ing to deepened starting point, shaft speed and coolant defini-tion	103

4.2 CENTERING (Cycle 240, ISO: G240, software option 19)

Cycle run

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface

2   The tool is centered at the programmed feed rate **F** to the programmed centering diameter or centering depth.

3   If defined, the tool remains at the centering depth.

4   Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

ⓘ   Program a positioning block for the starting point (hole center) in the working plane with the radius compensation **R0**

The algebraic sign for the **Q344** (diameter) or **Q201** (depth) cycle parameter determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶   Enter depth as negative

▶   Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range: 0 to 99999.9999

▶ **Q343 Select diameter/depth (1/0)**: Select whether centering is based on the entered diameter or depth. If the control is to center based on the entered diameter, the point angle of the tool must be defined in the **T-angle** column of the TOOL.T tool table.
0: Centering based on the entered depth
1: Centering based on the entered diameter

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined. Input range: -99999.9999 to 99999.9999

▶ **Q344 Diameter of counterbore** (algebraic sign): Centering diameter. Only effective if Q343=1 is defined. Input range: -99999.9999 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during centering. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**

▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
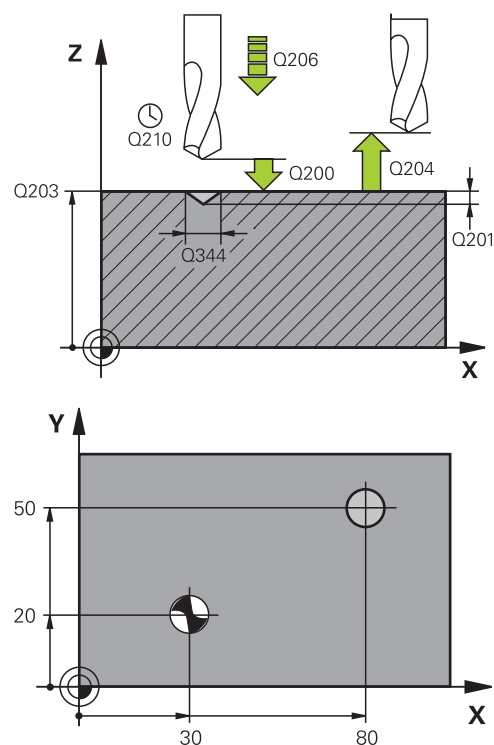
Example

10 L Z+100 R0 FMAX	
11 CYCL DEF 240 CENTERING	
Q200=2	;SET-UP CLEARANCE
Q343=1	;SELECT DIA./DEPTH
Q201=+0	;DEPTH
Q344=-9	;DIAMETER
Q206=250	;FEED RATE FOR PLNGNG
Q211=0.1	;DWELL TIME AT DEPTH
Q203=+20	;SURFACE COORDINATE
Q204=100	;2ND SET-UP CLEARANCE
12 L X+30 Y+20 R0 FMAX M3 M99	
13 L X+80 Y+50 R0 FMAX M99	

4.3   DRILLING (Cycle 200)

Cycle run

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

2   The tool drills to the first plunging depth at the programmed feed rate **F**

3   The Control retracts the tool at **FMAX** to the set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to the set-up clearance above the first plunging depth.

4   The tool then drills deeper by the plunging depth at the programmed feed rate F.

5   The control repeats this process (2 to 4) until the programmed depth is reached (the dwell time from Q211 is effective with every infeed)

6   Finally, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you want to drill without chip breaking, make sure to define, in the **Q202** parameter, a higher value than the depth **Q201** plus the calculated depth based on the point angle. You can enter a much higher value there.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶   Enter depth as negative

▶   Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

Cycle parameters

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range: 0 to 99999.9999
- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:
  - ▪ the plunging depth is equal to the depth
  - ▪ the plunging depth is greater than the depth
- ▶ **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal by the Control. Input range: 0 to 3600.0000
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the TOOL.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool
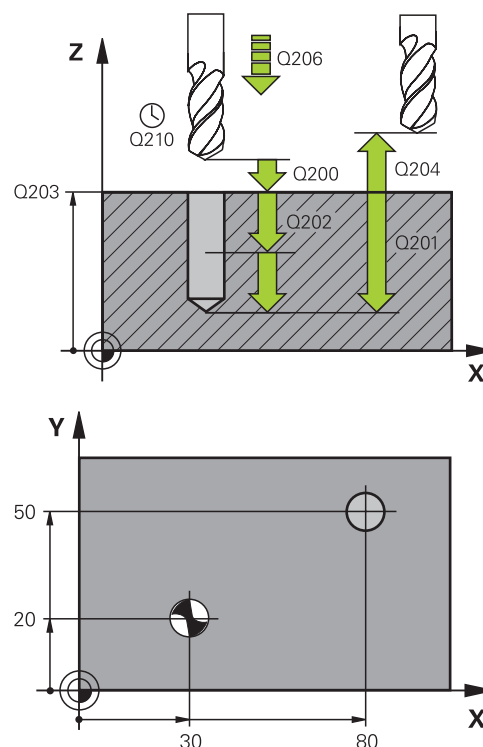
Example

11 CYCL DEF 200 DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-15	;DEPTH
Q206=250	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q211=0	;DWELL TIME AT TOP
Q203=+20	;SURFACE COORDINATE
Q204=100	;2ND SET-UP CLEARANCE
Q211=0.1	;DWELL TIME AT DEPTH
Q395=0	;DEPTH REFERENCE
12 L X+30 Y+20 FMAX M3	
13 CYCL CALL	
14 L X+80 Y+50 FMAX M99	

4.4 REAMING (Cycle 201, ISO: G201, software option 19)

Cycle run

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

2   The tool reams to the entered depth at the programmed feed rate **F**.

3   If programmed, the tool remains at the hole bottom for the entered dwell time.

4   Then, the control retracts the tool at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶   Enter depth as negative

▶   Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

Cycle parameters

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during reaming. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**

▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000

▶ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for reaming applies. Input range: 0 to 99999.999

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: 0 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
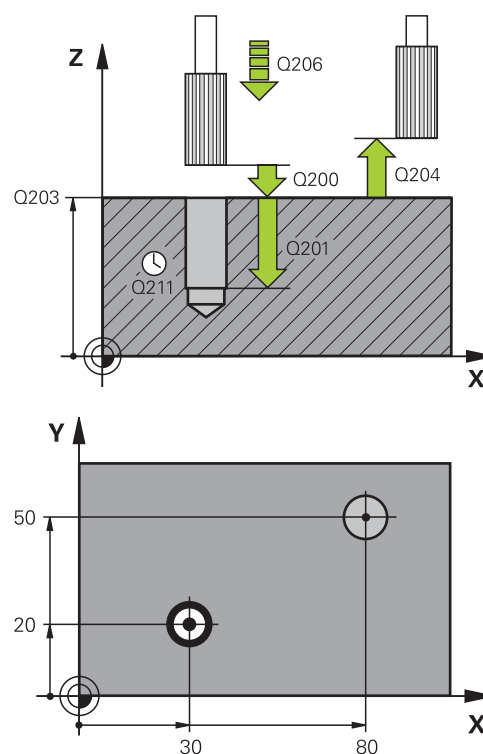
Example

11 CYCL DEF 201 REAMING	
Q200=2	;SET-UP CLEARANCE
Q201=-15	;DEPTH
Q206=100	;FEED RATE FOR PLNGNG
Q211=0.5	;DWELL TIME AT DEPTH
Q208=250	;RETRACTION FEED RATE
Q203=+20	;SURFACE COORDINATE
Q204=100	;2ND SET-UP CLEARANCE
12 L X+30 Y+20 FMAX M3	
13 CYCL CALL	
14 L X+80 Y+50 FMAX M9	
15 L Z+100 FMAX M2	

4.5 BORING (Cycle 202, ISO: G202, software option 19)

Cycle run

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface

2   The tool drills to the programmed depth at the feed rate for plunging.

3   If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.

4   The control then carries out an oriented spindle stop to the position that is defined in the **Q336** parameter

5   If retraction is selected, the control retracts in the programmed direction by 0.2 mm (fixed value)

6   The tool then retracts to set-up clearance at the retraction rate, and from there —if programmed—to the 2nd set-up clearance at **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**. If **Q214**=0 the tool tip remains on the wall of the hole

7   The control then returns the tool to the center of the hole

Please note while programming:

⚙   Machine and control must be specially prepared by the machine tool builder for use of this cycle.

This cycle can only be used on machines with a servo-controlled spindle.

ℹ   Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

After machining, the control returns the tool to the starting point of the working plane. This way, you can continue positioning the tool incrementally.

If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

NOTICE

Danger of collision!

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

▶ Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.

▶ Select the angle so that the tool tip is parallel to the disengaging direction

▶ Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters

▸ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

▸ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999

▸ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during boring. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**

▸ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000

▸ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for plunging applies. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**

▸ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▸ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▸ **Q214 Disengaging directn (0/1/2/3/4)?**: Determine the direction in which the control retracts the tool at the hole bottom (after carrying out an oriented spindle stop)
0: Do not disengage the tool
1: Disengage the tool in the minus direction of the reference axis
2: Disengage the tool in the minus direction of the minor axis
3: Disengage the tool in the plus direction of the reference axis
4: Disengage the tool in the plus direction of the minor axis

▸ **Q336 Angle for spindle orientation?** (absolute): Angle to which the control positions the tool before retracting it. Input range: -360.000 to 360.000
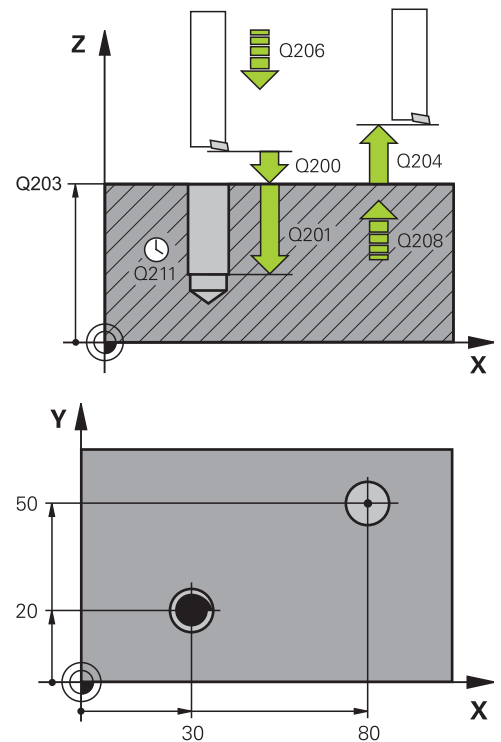
Example

10 L Z+100 R0 FMAX	
11 CYCL DEF 202 BORING	
Q200=2	;SET-UP CLEARANCE
Q201=-15	;DEPTH
Q206=100	;FEED RATE FOR PLNGNG
Q211=0.5	;DWELL TIME AT DEPTH
Q208=250	;RETRACTION FEED RATE
Q203=+20	;SURFACE COORDINATE
Q204=100	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE
12 L X+30 Y+20 FMAX M3	
13 CYCL CALL	
14 L X+80 Y+50 FMAX M99	

4.6 UNIVERSAL DRILLING (Cycle 203, ISO: G203, software option 19)

Cycle run

Behavior without chip breaking and without decrement:

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface

2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**

3 Then, the control retracts the tool from the hole to **SET-UP CLEARANCE Q200**

4 Now, the control again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTHQ202 FEED RATE FOR PLNGNG Q206**

5 When machining without chip breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and remains there for the **DWELL TIME AT TOP Q210**.

6 This procedure is repeated until **depth Q201** is achieved.

7 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and without decrement:

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface

2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**

3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**

4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** at **FEED RATE FOR PLNGNG Q206**

5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**

6 If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**

7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached

8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached.

9 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and with decrement

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified **SAFETY CLEARANCE Q200** above the workpiece surface

2   The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**

3   Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**

4   Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** minus **DECREMENT Q212** at **FEED RATE FOR PLNGNG Q206**. The increasingly smaller difference between the updated **PLUNGING DEPTH Q202** minus **DECREMENT Q212** must never be smaller than **MIN. PLUNGING DEPTH Q205** (example: **Q202**=5, **Q212**=1, **Q213**=4, **Q205**= 3: The first plunging depth is 5 mm, the second plunging depth is 5 - 1 = 4 mm, the third plunging depth is 4 - 1 = 3 mm, the fourth plunging depth is also 3 mm)

5   The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**

6   If programmed, the control will now wait for the time specified in **DWELL TIME AT TOP Q210**

7   Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached

8   Steps 2 to 7 will be repeated until **DEPTH Q201** is reached.

9   If programmed, the control will now wait for the time specified in **DWELL TIME AT DEPTH Q211**

10   When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Please note while programming:

> **i** Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
>
> The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999

- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**

- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999

    The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:
    - ▪ the plunging depth is equal to the depth
    - ▪ the plunging depth is greater than the depth

- ▶ **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal by the Control. Input range: 0 to 3600.0000

- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

- ▶ **Q212 Decrement?** (incremental): Value by which the control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999

- ▶ **Q213 Nr of breaks before retracting?**: Number of chip breaks before the control will retract the tool from the hole for chip removal. For chip breaking, the control retracts the tool each time by the value in **Q256**. Input range: 0 to 99999

- ▶ **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999
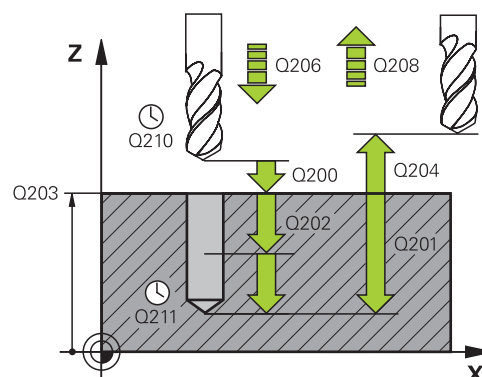
Example

11 CYCL DEF 203 UNIVERSAL DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q211=0	;DWELL TIME AT TOP
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q212=0.2	;DECREMENT
Q213=3	;NR OF BREAKS
Q205=3	;MIN. PLUNGING DEPTH
Q211=0.25	;DWELL TIME AT DEPTH
Q208=500	;RETRACTION FEED RATE
Q256=0.2	;DIST FOR CHIP BRKNG
Q395=0	;DEPTH REFERENCE

▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000

▶ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the control retracts the tool at the feed rate specified in **Q206**. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**

▶ **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control retracts the tool during chip breaking. Input range 0.000 to 99999.999

▶ **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the TOOL.T tool table.
0 = Depth references the tool tip
1 = Depth references the cylindrical part of the tool

4.7 BACK BORING (Cycle 204, ISO: G204, software option 19)

Cycle run

This cycle allows counterbores to be machined from the underside of the workpiece.

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface

2 The control then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.

3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the cutting edge has reached programmed set-up clearance beneath the lower workpiece edge

4 The control then centers the tool again in the bore hole, switches on the spindle and the coolant and moves at the feed rate for counterboring to the depth programmed for the counterbore

5 If programmed, the tool remains at the counterbore bottom. The tool will then be retracted from the hole again. The control carries out another oriented spindle stop and the tool is once again displaced by the off-center distance

6 Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

7 The control then returns the tool to the center of the hole
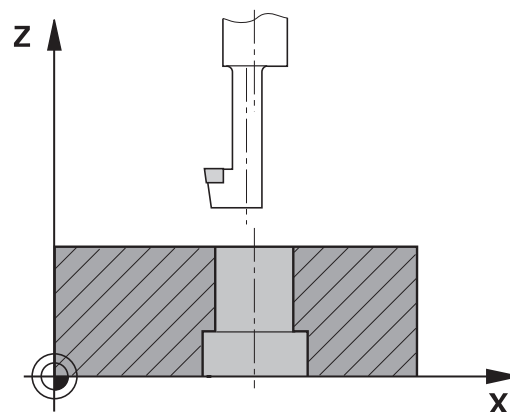
Please note while programming:

⚙️ Machine and control must be specially prepared by the machine tool builder for use of this cycle.

This cycle can only be used on machines with a closed-loop spindle.

Special boring bars for upward cutting are required for this cycle.

ℹ️ Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

After machining, the control returns the tool to the starting point of the working plane. This way, you can continue positioning the tool incrementally.

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

Enter the tool length so that the lower edge of the boring bar is measured, not the cutting edge.

When calculating the starting point for boring, the control considers the cutting edge length of the boring bar and the thickness of the material.

If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

NOTICE

Danger of collision!

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

▶ Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.

▶ Select the angle so that the tool tip is parallel to the disengaging direction

▶ Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

- **Q249 Depth of counterbore?** (incremental): Distance between underside of workpiece and bottom of the hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range: -99999.9999 to 99999.9999

- **Q250 Material thickness?** (incremental): Thickness of the workpiece. Input range: 0.0001 to 99999.9999

- **Q251 Tool edge off-center distance?** (incremental): Off-center distance for the boring bar; value from the tool data sheet. Input range: 0.0001 to 99999.9999

- **Q252 Tool edge height?** (incremental): Distance between the underside of the boring bar and the main cutting edge; value from tool data sheet. Input range: 0.0001 to 99999.9999

- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**

- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**

- **Q255 Dwell time in secs.?**: Dwell time in seconds at the bottom of the bore hole. Input range: 0 to 3600.000

- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
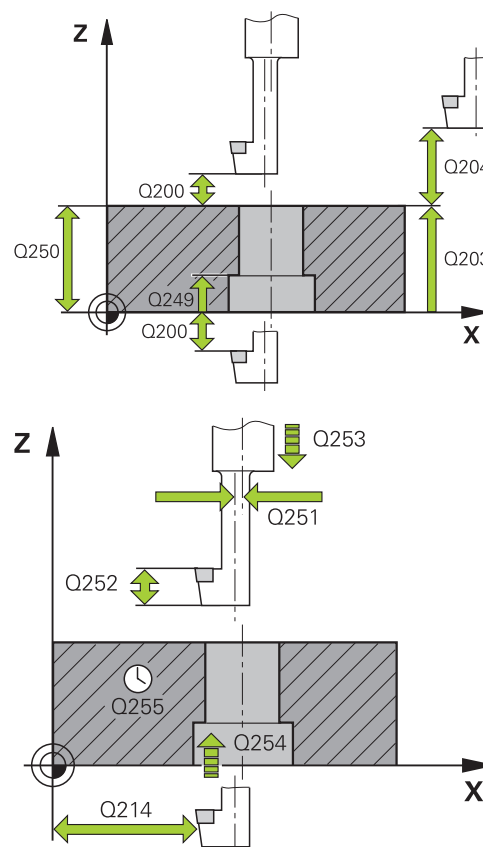
Example

11 CYCL DEF 204 BACK BORING	
Q200=2	;SET-UP CLEARANCE
Q249=+5	;DEPTH OF COUNTERBORE
Q250=20	;MATERIAL THICKNESS
Q251=3.5	;OFF-CENTER DISTANCE
Q252=15	;TOOL EDGE HEIGHT
Q253=750	;F PRE-POSITIONING

▶ **Q214 Disengaging directn (0/1/2/3/4)?**:
Determine the direction in which the control
will displace the tool by the off-center distance
(after having carried out an oriented spindle stop);
programming 0 is not allowed
1: Retract the tool in the negative direction of the
reference axis
2: Retract the tool in the negative direction of the
minor axis
3: Retract the tool in the positive direction of the
reference axis
4: Retract the tool in the positive direction of the
minor axis

▶ **Q336 Angle for spindle orientation?** (absolute):
Angle at which the control positions the tool
before it is plunged into or retracted from the bore
hole. Input range: -360.0000 to 360.0000

Q254=200	;F COUNTERBORING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE

4.8 UNIVERSAL PECKING (Cycle 205, ISO: G205, software option 19)

Cycle run

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

2   If you enter a deepened starting point, the control moves at the defined positioning feed rate to the set-up clearance above the recessed starting point

3   The tool drills to the first plunging depth at the programmed feed rate **F**

4   If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered advanced stop distance above the first plunging depth

5   The tool then drills deeper by the plunging depth at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.

6   The control repeats this procedure (steps 2 to 4) until the total hole depth is reached

7   The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to set-up clearance or the 2nd set-up clearance at the retraction feed rate. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

ℹ Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you enter advance stop distances **Q258** not equal to **Q259**, the control will change the advance stop distances between the first and last plunging depths at the same rate.

If you use **Q379** to enter a recessed starting point, the control will change the starting point of the infeed movement. Retraction movements are not changed by the control, they are always calculated with respect to the coordinate of the workpiece surface.


NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range: -99999.9999 to 99999.9999

- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**

- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:

  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

- ▶ **Q212 Decrement?** (incremental): Value by which the control decreases the **Q202** plunging depth. Input range: 0 to 99999.9999

- ▶ **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999

- ▶ **Q258 Upper advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole. Input range 0 to 99999.9999

- ▶ **Q259 Lower advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole; value for the last plunging depth. Input range: 0 to 99999.9999
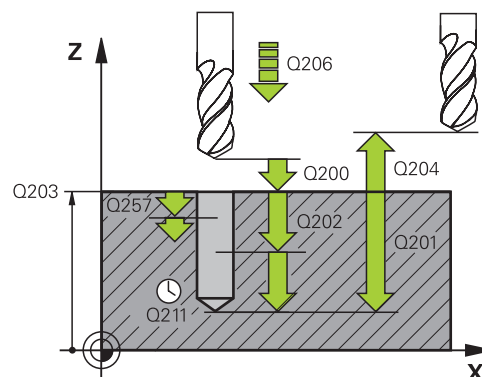
Example

11 CYCL DEF 205 UNIVERSAL PECKING	
Q200=2	;SET-UP CLEARANCE
Q201=-80	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=15	;PLUNGING DEPTH
Q203=+100	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q212=0.5	;DECREMENT
Q205=3	;MIN. PLUNGING DEPTH
Q258=0.5	;UPPER ADV STOP DIST
Q259=1	;LOWER ADV STOP DIST
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=0.2	;DIST FOR CHIP BRKNG
Q211=0.25	;DWELL TIME AT DEPTH
Q379=7.5	;STARTING POINT
Q253=750	;F PRE-POSITIONING
Q208=9999	;RETRACTION FEED RATE
Q395=0	;DEPTH REFERENCE

▶ **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999

▶ **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control retracts the tool during chip breaking. Input range 0.000 to 99999.999

▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000

▶ **Q379 Deepened starting point?** (incremental, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed starting point. Input range: 0 to 99999.9999

▶ **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**

▶ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q206**. Input range: 0 to 99999,9999; alternatively **FMAX, FAUTO**

▶ **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the TOOL.T tool table.
0 = Depth references the tool tip
1 = Depth references the cylindrical part of the tool

Position behavior when working with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Start of drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,2 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2

- The starting point of drilling is calculated as follows: 0.2 x Q379=0.2*2=0.4; the starting point is 0.4 mm/inch above the recessed starting point. So if the recessed starting point is at -2, the control starts the drilling process at -1.6 mm

The following table shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.2 * Q379	Start of drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (Q200=2, 5>2, thus the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (Q200=2, 20>2, thus the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (Q200=5, 20>5, thus the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip breaking

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

■ The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,8 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

Example:

■ **SURFACE COORDINATE Q203** =0

■ **SET-UP CLEARANCEQ200** =2

■ **STARTING POINT Q379** =2

■ The position for chip removal is calculated as follows: 0.8 x Q379 = 0.8 * 2 = 1.6; the position for chip removal is 1.6 mm/inch above the recessed start point. So if the recessed starting point is at -2, the control starts chip removal at -0.4

The following table shows various examples for calculating the position for chip breaking (retraction position):

Position for chip breaking (retraction position) with deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (Q200=2, 8>2, thus the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (Q200=2, 20>2, thus the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (Q200=2, 80>2, thus the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (Q200=5, 8>5, thus the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (Q200=5, 20>5, thus the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (Q200=5, 80>5, thus the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (Q200=20, 80>20, thus the value 20 is used.)	-80

4.9 BORE MILLING (Cycle 208, Software option 19)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface. Then, it moves the tool on a circular arc with the indicated diameter (provided that enough space is available)

2 The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate **F**.

3 When the drilling depth is reached, the control once again traverses a full circle to remove the material remaining after the initial plunge.

4 The control then positions the tool at the center of the hole again.

5 Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

ℹ️ 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 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.

Note that if the infeed distance is too large, the tool or the workpiece may be damaged.

To prevent the infeeds from being too large, enter the maximum plunge angle of the tool in the **ANGLE** column of the tool table. The control then automatically calculates the max. infeed permitted and changes your entered value accordingly.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during helical drilling. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q334 Feed per revolution of helix** (incremental): Depth of the tool plunge with each helix (=360°). Input range: 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▶ **Q335 Nominal diameter?** (absolute): 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. Input range: 0 to 99999.9999

▶ **Q342 Roughing diameter?** (absolute): As soon as you enter a value greater than 0 in Q342, the control no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter. Input range: 0 to 99999.9999

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
+1 = Climb
-1 = Up-cut (if you enter 0, climb milling is performed)
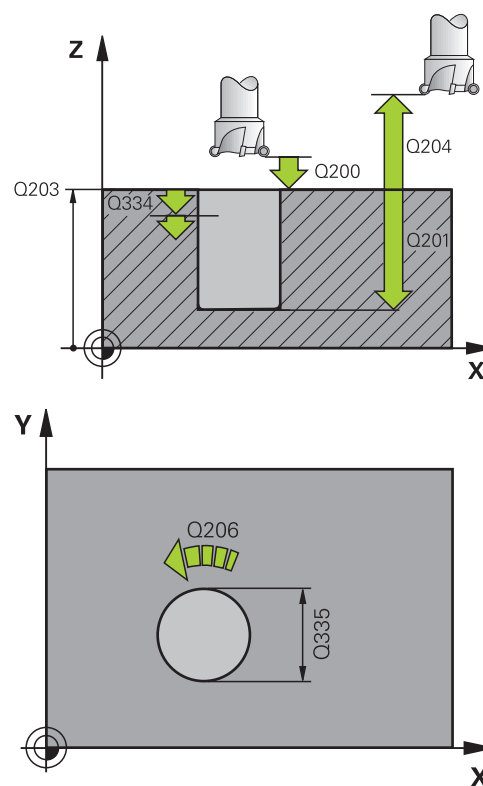
Example

12 CYCL DEF 208 BORE MILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-80	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q334=1.5	;PLUNGING DEPTH
Q203=+100	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q335=25	;NOMINAL DIAMETER
Q342=0	;ROUGHING DIAMETER
Q351=+1	;CLIMB OR UP-CUT

4.10 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, ISO: G241, software option 19)

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **Safety clearance Q200** above the workpiece **SURFACE COORDINATE Q203**

2. Depending on the "Position behavior when working with Q379", Page 96, the control will either switch on the spindle with the programmed speed at the **Safety clearance Q200** or at a certain distance above the coordinate surface. see Page 96

3. The control executes the approach motion depending on the direction of rotation defined in the cycle with a spindle that rotates clockwise, counterclockwise, or is stationary

4. The tool drills to the hole depth at the feed rate **F**, or to the maximum plunging depth if a smaller infeed value has been entered. The plunging depth is decreased after each infeed by the decrement. If you have entered a dwell depth, the control reduces the feed rate by the feed rate factor after the dwell depth has been reached

5. If programmed, the tool remains at the hole bottom for chip breaking.

6. The control repeats this procedure (steps 4 to 5) until the total hole depth is reached

7. After the control has reached this position, it will automatically switch off the coolant as soon as the speed has reached the value defined in Q427 **ROT.SPEED INFEED/OUT**

8. The control positions the tool to the retract position at the retraction feed rate. To find out the retract position value in your particular case, please refer to: see Page 96

9. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

Please note while programming:

> **ℹ** Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
>
> The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and **Q203 SURFACE COORDINATE**. Input range: 0 to 99999.9999

▶ **Q201 Depth?** (incremental): Distance between **Q203 SURFACE COORDINATE** and bottom of hole. Input range: -99999.9999 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**

▶ **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000

▶ **Q203 Workpiece surface coordinate?** (absolute): Distance to workpiece datum. Input range: -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▶ **Q379 Deepened starting point?** (incremental, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed starting point. Input range: 0 to 99999.9999

▶ **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**

▶ **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208**=0, the control retracts the tool at **Q206 FEED RATE FOR PLNGNG**. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**

▶ **Q426 Rot. dir. of entry/exit (3/4/5)?**: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input:
3: Turn the spindle with M3
4: Turn the spindle with M4
5: Move with stationary spindle

▶ **Q427 Spindle speed of entry/exit?**: Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input range: 0 to 99999

▶ **Q428 Spindle speed for drilling?**: Desired speed for drilling. Input range: 0 to 99999
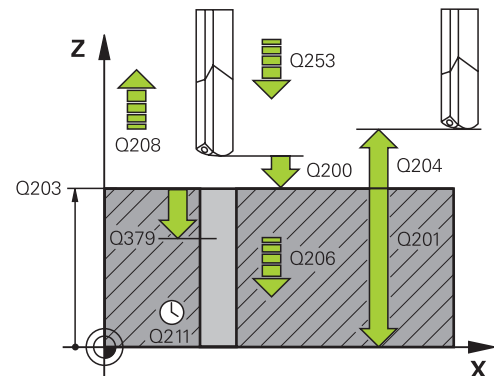
Example

11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG	
Q200=2	;SET-UP CLEARANCE
Q201=-80	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q211=0.25	;DWELL TIME AT DEPTH
Q203=+100	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q379=7.5	;STARTING POINT
Q253=750	;F PRE-POSITIONING
Q208=1000	;RETRACTION FEED RATE
Q426=3	;DIR. OF SPINDLE ROT.
Q427=25	;ROT.SPEED INFEED/OUT
Q428=500	;ROT. SPEED DRILLING
Q429=8	;COOLANT ON
Q430=9	;COOLANT OFF
Q435=0	;DWELL DEPTH
Q401=100	;FEED RATE FACTOR
Q202=9999	;MAX. PLUNGING DEPTH
Q212=0	;DECREMENT
Q205=0	;MIN. PLUNGING DEPTH

▶ **Q429 M function for coolant on?**: Miscellaneous function for switching on the coolant. The control switches on the coolant on if the tool is in the hole at **Q379 STARTING POINT**. Input range: 0 to 999

▶ **Q430 M function for coolant off?**: Miscellaneous function for switching off the coolant. The control switches the coolant off if the tool is at **Q201 DEPTH**. Input range: 0 to 999

▶ **Q435 Dwell depth?** (incremental): Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (default setting). Application: During machining of through-holes some tools require a short dwell time before leaving the bottom of the hole in order to transport the chips to the top. Define a value smaller than **Q201 DEPTH**; input range: 0 to 99999.9999

▶ **Q401 Feed rate factor in %?**: Factor by which the control reduces the feed rate after **Q435 DWELL DEPTH** has been reached. Input range: 0 to 100

▶ **Q202 Maximum plunging depth?** (incremental): Infeed per cut. **Q201 DEPTH** does not have to be a multiple of **Q202**. Input range 0 to 99999.9999

▶ **Q212 Decrement?** (incremental): Value by which the control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999

▶ **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999

Position behavior when working with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Start of drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

  The starting point is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,2 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

  Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2
- The starting point of drilling is calculated as follows:
  0.2 x Q379=0.2*2=0.4; the starting point is 0.4 mm/inch above the recessed starting point. So if the recessed starting point is at -2, the control starts the drilling process at -1.6 mm

  The following table shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.2 * Q379	Start of drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (Q200=2, 5>2, thus the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (Q200=2, 20>2, thus the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (Q200=5, 20>5, thus the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip breaking

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point Q379. This value can be calculated as follows: **0,8 x Q379**; if the result of this calculation is larger than Q200, the value is always Q200.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2
- The position for chip removal is calculated as follows:
  0.8 x Q379 = 0.8 * 2 = 1.6; the position for chip removal is 1.6 mm/inch above the recessed start point. So if the recessed starting point is at -2, the control starts chip removal at -0.4

  The following table shows various examples for calculating the position for chip breaking (retraction position):

Position for chip breaking (retraction position) with deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (Q200=2, 8>2, thus the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (Q200=2, 20>2, thus the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (Q200=2, 80>2, thus the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (Q200=5, 8>5, thus the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (Q200=5, 20>5, thus the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (Q200=5, 80>5, thus the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (Q200=20, 80>20, thus the value 20 is used.)	-80

4.11   Programming Examples

Example: Drilling cycles
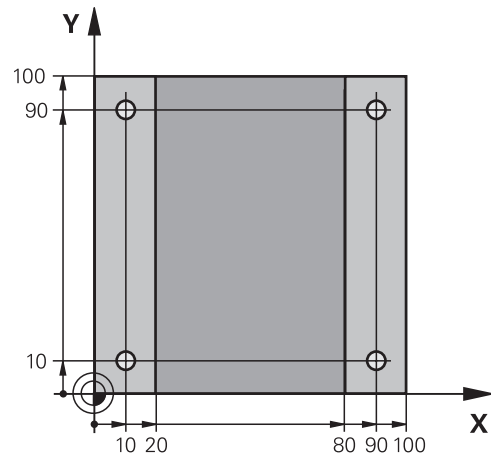
0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4500	Tool call (tool radius 3)
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 L X+10 Y+10 R0 FMAX M3	Approach hole 1, spindle ON
7 CYCL CALL	Cycle call
8 L Y+90 R0 FMAX M99	Approach hole 2, cycle call
9 L X+90 R0 FMAX M99	Approach hole 3, cycle call
10 L Y+10 R0 FMAX M99	Approach hole 4, cycle call
11 L Z+250 R0 FMAX M2	Retract the tool, end program
12 END PGM C200 MM	

Example: Using drilling cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the pattern definition PATTERN DEF POS and are called by the Control with CYCL CALL PAT.

The tool radii have been selected in such a way that all work steps can be seen in the test graphics.

Program sequence

- Centering (tool radius 4)
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)
  Further information: "Fundamentals", Page 116

0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL CALL 1 Z S5000	Tool call: centering tool (tool radius 4)
4 L Z+50 R0 FMAX	Move tool to clearance height
5 PATTERN DEF	Define all drilling positions in the point pattern
POS1(X+10 Y+10 Z+0)	
POS2(X+40 Y+30 Z+0)	
POS3(X+20 Y+55 Z+0)	
POS4(X+10 Y+90 Z+0)	
POS5(X+90 Y+90 Z+0)	
POS6(X+80 Y+65 Z+0)	
POS7(X+80 Y+30 Z+0)	
POS8(X+90 Y+10 Z+0)	
6 CYCL DEF 240 CENTERING	Cycle definition: centering
Q200=2 ;SET-UP CLEARANCE	
Q343=0 ;SELECT DIA./DEPTH	
Q201=-2 ;DEPTH	
Q344=-10 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
POSITION 7 GLOBAL DEF 125	This function is used for CYCL CALL PAT and positions the tool at the 2nd set-up clearance between the points. This function remains active until M30 is executed.
Q345=+1 ;SELECT POS. HEIGHT	
7 CYCL CALL PAT F5000 M13	Cycle call in connection with the point pattern

8 L Z+100 R0 FMAX		Retract the tool
9 TOOL CALL 2 Z S5000		Tool call: drill (radius 2.4)
10 L Z+50 R0 F5000		Move tool to clearance height
11 CYCL DEF 200 DRILLING		Cycle definition: drilling
Q200=2	;SET-UP CLEARANCE	
Q201=-25	;DEPTH	
Q206=150	;FEED RATE FOR PLNGNG	
Q202=5	;PLUNGING DEPTH	
Q211=0	;DWELL TIME AT TOP	
Q203=+0	;SURFACE COORDINATE	
Q204=10	;2ND SET-UP CLEARANCE	
Q211=0.2	;DWELL TIME AT DEPTH	
Q395=0	;DEPTH REFERENCE	
12 CYCL CALL PAT F500 M13		Cycle call in connection with the point pattern
13 L Z+100 R0 FMAX		Retract the tool
14 TOOL CALL Z S200		Tool call: tap (radius 3)
15 L Z+50 R0 FMAX		Move tool to clearance height
16 CYCL DEF 206 TAPPING		Cycle definition: tapping
Q200=2	;SET-UP CLEARANCE	
Q201=-25	;DEPTH OF THREAD	
Q206=150	;FEED RATE FOR PLNGNG	
Q211=0	;DWELL TIME AT DEPTH	
Q203=+0	;SURFACE COORDINATE	
Q204=10	;2ND SET-UP CLEARANCE	
17 CYCL CALL PAT F5000 M13		Cycle call in connection with the point pattern
18 L Z+100 R0 FMAX M2		Retract the tool, end program
19 END PGM 1 MM		

5

Fixed Cycles:
Tapping / Thread
Milling

5.1   Fundamentals

Overview

The control offers the following cycles for all types of threading operations:

Soft key	Cycle	Page
206	206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	117
207 RT	207 TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	120
209 RT	209 TAPPING WITH CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	125
262	262 THREAD MILLING Cycle for milling a thread in pre-drilled material	132
263	263 THREAD MILLING/ COUNTERSINKING Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	136
264	264 THREAD DRILLING/ MILLING Cycle for drilling into solid material with subsequent milling of the thread with a tool	140
265	265 HELICAL THREAD DRILLING/MILLING Cycle for milling the thread into solid material	144
267	267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer	148

5.2 TAPPING with a floating tap holder (Cycle 206, ISO: G206)

Cycle run

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

2   The tool drills to the total hole depth in one movement.

3   Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

4   At the set-up clearance, the direction of spindle rotation reverses once again.

Please note while programming:

ℹ   Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (feed rate override is not active). The Control then adapts the spindle speed as required.

- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.

- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

The spindle speed potentiometer is inactive.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message. In Cycle 206, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

Guide value: 4x pitch.

▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during tapping. Input range 0 to 99999.999 alternatively **FAUTO**

▶ **Q211 Dwell time at the depth?**: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
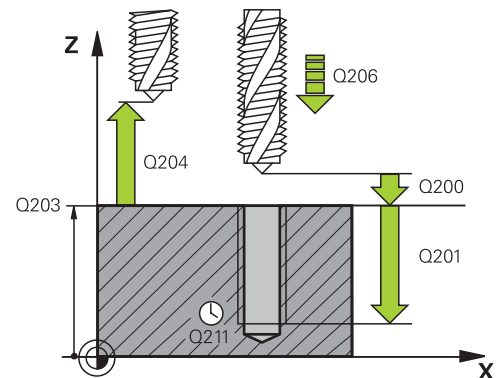
Example

25 CYCL DEF 206 TAPPING NEU	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q206=150	;FEED RATE FOR PLNGNG
Q211=0.25	;DWELL TIME AT DEPTH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

The feed rate is calculated as follows: F = S x p

F: Feed rate (mm/min)

S: Spindle speed (rpm)

p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the **NC Stop** key, the control will display a soft key with which you can retract the tool.

5.3 TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)

Cycle run

The control cuts the thread without a floating tap holder in one or more passes.

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

2   The tool drills to the total hole depth in one movement.

3   It then reverses the direction of spindle rotation and the tool is retracted to the set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

4   The control stops the spindle turning at that set-up clearance

Please note while programming:

> ⚙ Machine and control must be specially prepared by the machine tool builder for use of this cycle.
>
> This cycle is effective only for machines with servo-controlled spindle.

Fixed Cycles: Tapping / Thread Milling | TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)

5

ⓘ Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (speed override is not active). The Control then adapts the spindle speed as required.
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
- **limitSpindleSpeed** (no. 113604): Spindle speed limit True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time False: (Limiting not active)

The spindle speed potentiometer is inactive.

If you program M3 (or M4) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the TOOL CALL block).

If you do not program M3 (or M4) before this cycle, the spindle stands still after the end of the cycle. Then you must restart the spindle with M3 (or M4) before the next operation.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.

For tapping, the spindle and the tool axis are always synchronized with each other. The synchronization can be carried out while the spindle is rotating or while it is stationary.

If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis leaves the acceleration path within this distance.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

Fixed Cycles: Tapping / Thread Milling | TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)

5

Cycle parameters

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- ▶ **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  − = left-hand thread
  Input range: -99.9999 to +99.9999
- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

Example

26 CYCL DEF 207 RIGID TAPPING NEU	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q239=+1	;THREAD PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

Retracting after a program interruption

Retracting in the Manual Operation mode

You can interrupt the thread cutting process by pressing the **NC Stop** key. A soft key for retracting the tool from the thread is displayed in the lower soft-key row. When you press this soft key and the **NC Start** key, the tool retracts from the hole and returns to the starting point of machining. The spindle is stopped automatically. The control displays a message.

Retracting in the Program Run, Single Block or Full Sequence mode

You can interrupt the thread cutting process by pressing the **NC Stop** key. The control displays the **MANUAL TRAVERSE** soft key. After you pressed the **MANUAL TRAVERSE** soft key, you can retract the tool in the active spindle axis. To resume machining after the interruption, press the **RESTORE POSITION** soft key and **NC Start**. The control moves the tool back to the position it had assumed before the **NC Stop** key was pressed.

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

▶ When retracting the tool you can move it in the positive and negative tool axis directions

▶ Be aware of the direction in which you retract the tool from the hole before retracting

---

5.4 TAPPING WITH CHIP BREAKING (Cycle 209, ISO: G209, software option 19)

Cycle run

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 or not.

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 the 2nd set-up clearance at **FMAX**

6   The control stops the spindle turning at that set-up clearance

Please note while programming:

⚙ Machine and control must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

ℹ 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.

Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

- **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (feed rate override is not active). The Control then adapts the spindle speed as required.
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
- **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

The spindle speed potentiometer is inactive.

If you 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.

If you program M3 (or M4) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the TOOL CALL block).

If you do not program M3 (or M4) before this cycle, the spindle stands still after the end of the cycle. Then you must restart the spindle with M3 (or M4) before the next operation.

If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.

For tapping, the spindle and the tool axis are always synchronized with each other. The synchronization can be carried out while the spindle is rotating or while it is stationary.

If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis can leave the acceleration path within this distance

---

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

Cycle parameters

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999

- ▶ **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  −= left-hand thread
  Input range: -99.9999 to +99.9999

- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

- ▶ **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999

- ▶ **Q256 Retract dist. for chip breaking?**: The control multiplies the pitch **Q239** by the programmed value and retracts the tool by the calculated value for chip breaking. If you enter **Q256** = 0, the control retracts the tool completely from the hole (to the set-up clearance) for chip breaking. Input range 0.000 to 99999.999

- ▶ **Q336 Angle for spindle orientation?** (absolute): Angle to which the control positions the tool before machining the thread. This allows you to recut the thread, if required. Input range -360.0000 to 360.0000

- ▶ **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 hole. Input range 0.0001 to 10. Maximum increase to maximum speed of the active gear stage.
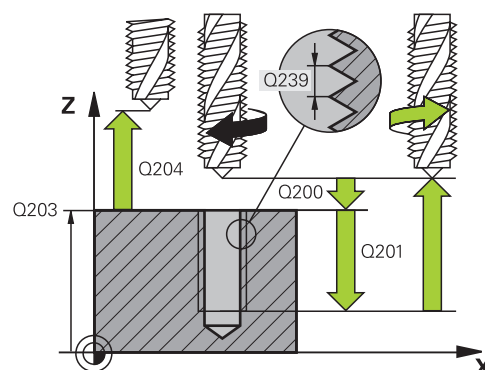
Example

26 CYCL DEF 209 TAPPING W/ CHIP BRKG	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH OF THREAD
Q239=+1	;THREAD PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=+1	;DIST FOR CHIP BRKNG
Q336=50	;ANGLE OF SPINDLE
Q403=1.5	;RPM FACTOR

Retracting after a program interruption

Retracting in the Manual Operation mode

You can interrupt the thread cutting process by pressing the **NC Stop** key. A soft key for retracting the tool from the thread is displayed in the lower soft-key row. When you press this soft key and the **NC Start** key, the tool retracts from the hole and returns to the starting point of machining. The spindle is stopped automatically. The control displays a message.

Retracting in the Program Run, Single Block or Full Sequence mode

You can interrupt the thread cutting process by pressing the **NC Stop** key. The control displays the **MANUAL TRAVERSE** soft key. After you pressed the **MANUAL TRAVERSE** soft key, you can retract the tool in the active spindle axis. To resume machining after the interruption, press the **RESTORE POSITION** soft key and **NC Start**. The control moves the tool back to the position it had assumed before the **NC Stop** key was pressed.

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

► When retracting the tool you can move it in the positive and negative tool axis directions
► Be aware of the direction in which you retract the tool from the hole before retracting

---

5.5   Fundamentals of Thread Milling

Prerequisites

- 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 program the compensation with the delta value for the tool radius **DR** in the **TOOL CALL**
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265 you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread / − = left-hand thread) and milling method 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 would like 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

▶ Change to Positioning with Manual Data Input mode of operation

▶ First move the tool in a linear movement towards the hole center

▶ Retract the tool in the tool axis direction

ℹ 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.

The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRROR IMAGE in only one axis.

5.6 THREAD MILLING (Cycle 262, ISO: G262, software option 19)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

2 The tool 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 the 2nd setup clearance
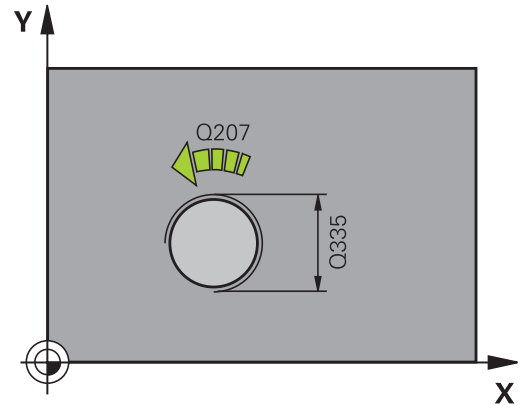
Please note while programming:

> **ℹ** 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 program the thread depth =0, the cycle will not be executed.
>
> 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 pitch of the tool diameter is four times smaller than the nominal thread diameter.
>
> Note that the control makes a compensating movement in the tool axis before the approach. The length of the compensation movement is at most half of the thread pitch. Ensure sufficient space in the hole!
>
> If you change the thread depth, the control will automatically move the starting point for the helical movement.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999

▶ **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+ = right-hand thread
–= left-hand thread
Input range: -99.9999 to +99.9999

▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999

▶ **Q355 Number of threads per step?**: Number of turns by which the tool is displaced:
0 = one helix over the entire thread depth
1 = continuous helix over the entire thread length
>1 = several helix paths with approach and departure, between these, the control offsets the tool by **Q355** x pitch. Input range 0 to 99999

▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
+1 = Climb
-1 = Up-cut (if you enter 0, climb milling is performed)

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **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 range 0 to 99999.999 alternatively **FAUTO**

Q355 = 0	Q355 = 1	Q355 > 1

Example

25 CYCL DEF 262 THREAD MILLING	
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
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q207=500	;FEED RATE FOR MILLNG
Q512=0	;FEED FOR APPROACH

5.7 THREAD MILLING/COUNTERSINKING (Cycle 263, ISO: G263, software option 19)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

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 the 2nd setup clearance

Please note while programming:

> **i** Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
>
> The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
> 1. Thread depth
> 2. Countersinking depth
> 3. Depth at front
>
> If you program one of the depth parameter 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.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999

▶ **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+ = right-hand thread
–= left-hand thread
Input range: -99.9999 to +99.9999

▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999

▶ **Q356 Countersinking depth?** (incremental): Distance between workpiece surface and tool tip. Input range -99999.9999 to 99999.9999

▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
+1 = Climb
-1 = Up-cut (if you enter 0, climb milling is performed)

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

▶ **Q357 Safety clearance to the side?** (incremental): Distance between the cutting edge and the wall of the hole. Input range 0 to 99999.9999

▶ **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999

▶ **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▶ **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **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 range 0 to 99999.999 alternatively **FAUTO**

Example

25 CYCL DEF 263 THREAD MLLNG/ CNTSNKG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-16	;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=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLNG
Q512=0	;FEED FOR APPROACH

5.8 THREAD DRILLING/MILLING (Cycle 264, ISO: G264, software option 19)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

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 moved at rapid traverse to the set-up clearance, and then 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 hole 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 the 2nd setup clearance

Please note while programming:

i Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Countersinking depth
3. Depth at front

If you program one of the depth parameter 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.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

- ▶ **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999
- ▶ **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  −= left-hand thread
  Input range: -99.9999 to +99.9999
- ▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- ▶ **Q356 Total hole depth?** (incremental): Distance between workpiece surface and hole bottom. Input range -99999.9999 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (if you enter 0, climb milling is performed)
- ▶ **Q202 Maximum plunging depth?** (incremental): Infeed per cut. **Q201 DEPTH** does not have to be a multiple of **Q202**. Input range 0 to 99999.9999

  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
- ▶ **Q258 Upper advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole. Input range 0 to 99999.9999
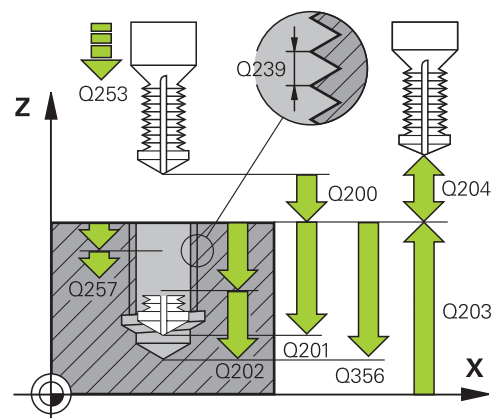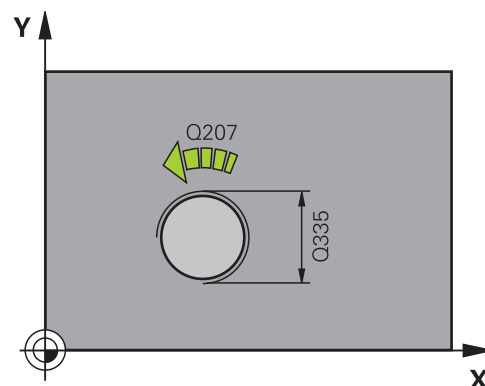
Example

25 CYCL DEF 264 THREAD DRILLNG/ MLLNG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-16	;DEPTH OF THREAD
Q356=-20	;TOTAL HOLE DEPTH
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT

▶ **Q257 Infeed depth for chip breaking?**
(incremental): Plunging depth after which the
control breaks the chip. No chip breaking if 0 is
entered. Input range 0 to 99999.9999

▶ **Q256 Retract dist. for chip breaking?**
(incremental): Value by which the control retracts
the tool during chip breaking. Input range 0.000 to
99999.999

▶ **Q358 Sinking depth at front?** (incremental):
Distance between tool point and the top surface
of the workpiece for countersinking at the front of
the tool. Input range -99999.9999 to 99999.9999

▶ **Q359 Countersinking offset at front?**
(incremental): Distance by which the control
moves the tool center away from the center. Input
range 0 to 99999.9999

▶ **Q200 Set-up clearance?** (incremental): Distance
between tool tip and workpiece surface. Input
range 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute):
Coordinate of the workpiece surface. Input range
-99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision
between tool and workpiece (fixtures) can occur.
Input range 0 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed
of the tool in mm/min during plunging. Input range
0 to 99999.999 alternatively **FAUTO**, **FU**

▶ **Q207 Feed rate for milling?**: Traversing speed of
the tool in mm/min during milling. Input range 0 to
99999.999 alternatively **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 range 0 to
99999.999 alternatively **FAUTO**

Q202=5	;PLUNGING DEPTH
Q258=0.2	;UPPER ADV STOP DIST
Q257=5	;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=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q206=150	;FEED RATE FOR PLNGNG
Q207=500	;FEED RATE FOR MILLNG
Q512=0	;FEED FOR APPROACH

5.9 HELICAL THREAD DRILLING/ MILLING (Cycle 265, ISO: G265, software option 19)

Cycle run

1   The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

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 the 2nd setup clearance

Please note while programming:

ⓘ Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign 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. Thread depth
2. Depth at front

If you program one of the depth parameter to be 0, the control does not execute that step.

If you change the thread depth, the control will automatically move the starting point for the helical movement.

The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999

▶ **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+ = right-hand thread
–= left-hand thread
Input range: -99.9999 to +99.9999

▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999

▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**

▶ **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999

▶ **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range 0 to 99999.9999

▶ **Q360 Countersink (before/after:0/1)?** :
Machining the chamfer
0 = before thread milling
1 = after thread milling

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**

Example

25 CYCL DEF 265 HEL. THREAD DRLG/ MLG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-16	;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=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLNG

5.10 EXTERNAL THREAD MILLING (Cycle 267, ISO: G267, software option 19)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

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 the 2nd setup clearance

Please note while programming:

ⓘ   Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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. Thread depth
2. Depth at front

If you program one of the depth parameter to be 0, the control does not execute that step.

The algebraic sign for the cycle parameter "thread depth" determines the working direction.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered
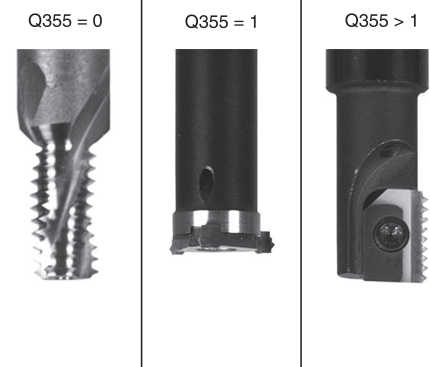
Cycle parameters

▶ **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999

▶ **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+ = right-hand thread
–= left-hand thread
Input range: -99.9999 to +99.9999

▶ **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999

▶ **Q355 Number of threads per step?**: Number of turns by which the tool is displaced:
0 = one helix over the entire thread depth
1 = continuous helix over the entire thread length
>1 = several helix paths with approach and departure, between these, the control offsets the tool by **Q355** x pitch. Input range 0 to 99999

▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
+1 = Climb
-1 = Up-cut (if you enter 0, climb milling is performed)

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

▶ **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999

▶ **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▶ **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **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 range 0 to 99999.999 alternatively **FAUTO**

Q355 = 0 Q355 = 1 Q355 > 1

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 FOR MILLNG
Q512=0	;FEED FOR APPROACH

5.11 Programming Examples

Example: Thread milling

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the control with **CYCL CALL PAT**.

The tool radii have been selected in such a way that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping

0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Tool call: centering tool
4 L Z+10 R0 F5000	Move tool to clearance height (program a value for F): the control positions the tool at the clearance height after every cycle
5 SEL PATTERN "TAB1"	Select the point table
6 CYCL DEF 240 CENTERING	Cycle definition: centering
Q200=2 ;SET-UP CLEARANCE	
Q343=1 ;SELECT DIA./DEPTH	
Q201=-3.5 ;DEPTH	
Q344=-7 ;DIAMETER	
Q206=150 ;FEED RATE FOR PLNGNG	
Q11=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
10 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT; feed rate between the points: 5000 mm/min
11 L Z+100 R0 FMAX M6	Retract the tool
12 TOOL CALL 2 Z S5000	Tool call: drill
13 L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	

Q210=0	;DWELL TIME AT TOP	
Q203=+0	;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0	;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2	;DWELL TIME AT DEPTH	
Q395=0	;DEPTH REFERENCE	
15 CYCL CALL PAT F5000 M3		Cycle call in connection with point table TAB1.PNT
16 L Z+100 R0 FMAX M6		Retract the tool
17 TOOL CALL 3 Z S200		Tool call: tap
18 L Z+50 R0 FMAX		Move tool to clearance height
19 CYCL DEF 206 TAPPING		Cycle definition: tapping
Q200=2	;SET-UP CLEARANCE	
Q201=-25	;DEPTH OF THREAD	
Q206=150	;FEED RATE FOR PLNGNG	
Q211=0	;DWELL TIME AT DEPTH	
Q203=+0	;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0	;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
20 CYCL CALL PAT F5000 M3		Cycle call in connection with point table TAB1.PNT
21 L Z+100 R0 FMAX M2		Retract the tool, end program
22 END PGM 1 MM		

TAB1. PNT point table

TAB1. PNTMM			
NR	X	Y	Z
0	+10	+10	+0
1	+40	+30	+0
2	+90	+10	+0
3	+80	+30	+0
4	+80	+65	+0
5	+90	+90	+0
6	+10	+90	+0
7	+20	+55	+0
[END]			

6

Fixed Cycles:
Pocket Milling /
Stud Milling /
Slot Milling

6.1 Fundamentals

Overview

The control offers the following cycles for machining pockets, studs and slots:

Soft key	Cycle	Page
	251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	157
	252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	163
	253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal plunging	170
	254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal plunging	175
	256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	181
	257 CIRCULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	186
	233 FACE MILLING Machining the face with up to 3 limits	196

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | RECTANGULAR POCKET (Cycle 251, ISO: G251, software option 19)

6

6.2 RECTANGULAR POCKET (Cycle 251, ISO: G251, software option 19)

Cycle run

Use Cycle 251, RECTANGULAR POCKET to completely machine rectangular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Roughing

1 The tool plunges the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter Q366.

2 The control roughs out the pocket from the inside out, taking the path overlap (parameter Q370) and the finishing allowance (parameters Q368 and Q369) into account.

3 At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall, then moves to the set-up clearance above the current pecking depth and returns from there at rapid traverse to the pocket center.

4 This process is repeated until the programmed pocket depth is reached.

Finishing

5 If finishing allowances have been defined, the control plunges and then approaches the contour. The 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

Please note while programming!

> **ℹ** With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.
>
> Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.
>
> Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).
>
> The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
>
> The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
>
> At the end of the cycle, the control returns the tool to the starting position.
>
> At the end of a roughing operation, the control returns the tool to the pocket center at rapid traverse. The tool is positioned at set-up clearance above the current plunging depth. Program a sufficient set-up clearance so that the tool cannot jam because of chips.
>
> The Control outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter (no. 201006).
>
> 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.

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | RECTANGULAR POCKET (Cycle 251, ISO: G251, software option 19)

6

### *NOTICE*

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

▶ Conduct a roughing operation beforehand
▶ Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

---

Cycle parameters

- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

- ▶ **Q218 First side length?** (incremental): Pocket length, parallel to the reference axis of the working plane Input range: 0 to 99999.9999

- ▶ **Q219 Second side length?** (incremental): Pocket length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999

- ▶ **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 range: 0 to 99999.9999

- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- ▶ **Q224 Angle of rotation?** (absolute): Angle by which the entire machining configuration is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: -360.0000 to 360.0000

- ▶ **Q367 Position of pocket (0/1/2/3/4)?**: Position of the pocket with respect to the position of the tool when the cycle is called:
  0: Tool position = pocket center
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner

- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | RECTANGULAR POCKET (Cycle 251, ISO: G251, software option 19)

6

- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999

- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

- ▶ **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1,9999; alternatively **PREDEF**

Example

8 CYCL DEF 251 RECTANGULAR POCKET	
Q215=0	;MACHINING OPERATION
Q218=80	;FIRST SIDE LENGTH
Q219=60	;2ND SIDE LENGTH
Q220=5	;CORNER RADIUS
Q368=0.2	;ALLOWANCE FOR SIDE
Q224=+0	;ANGLE OF ROTATION
Q367=0	;POCKET POSITION

▶ **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
0: vertical plunging. The control plunges the tool perpendicularly, regardless of the **ANGLE** plunging 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 generates an error message
2: reciprocating 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. The reciprocation length depends on the plunging angle. As a minimum value, the control uses twice the tool diameter
PREFEF: The control uses the value from the GLOBAL DEF block

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
0: Feed rate with respect to the tool center point path
1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
3: Feed rate always refers to the cutting edge

Q207=500	;FEED RATE FOR MILLNG
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
Q370=1	;TOOL PATH OVERLAP
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE

9 L X+50 Y+50 R0 FMAX M3 M99

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | CIRCULAR POCKET (Cycle 252, ISO: G252, software option 19)

6

6.3 CIRCULAR POCKET (Cycle 252, ISO: G252, software option 19)

Cycle run

Use Cycle 252 CIRCULAR POCKET 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

Roughing

1. The control first moves the tool at rapid traverse to the 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 (parameter Q370) and the finishing allowance (parameters 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 the 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 2nd 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 and 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 positions the tool again in the tool axis near the pocket wall at a distance corresponding to the finishing allowance Q368 and 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 the 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 the set-up clearance Q200 in the tool axis and returns at rapid traverse to the pocket center.

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | CIRCULAR POCKET (Cycle 252, ISO: G252, software option 19)

6

Please note while programming:

ℹ️ With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position (circle center) with radius compensation **R0**.

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the control returns the tool to the starting position.

At the end of a roughing operation, the control returns the tool to the pocket center at rapid traverse. The tool is positioned at set-up clearance above the current plunging depth. Program a sufficient set-up clearance so that the tool cannot jam because of chips.

The Control outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter (no. 201006).

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

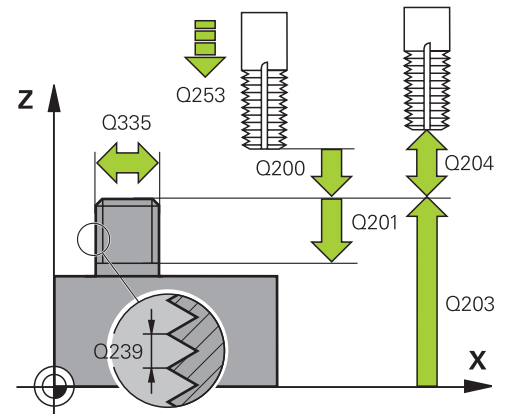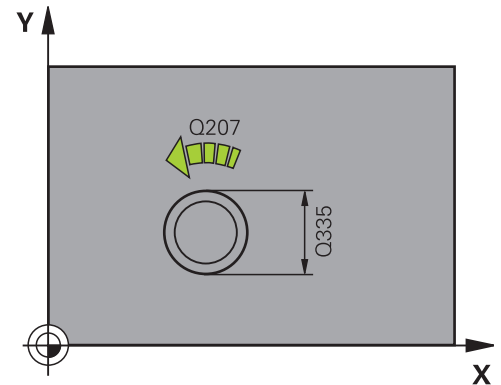### *NOTICE*

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered
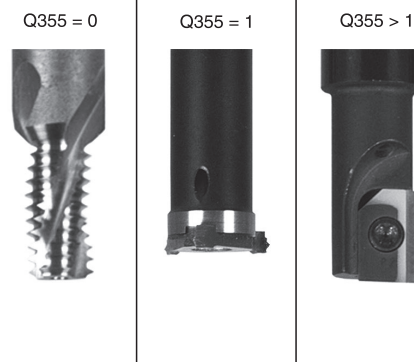

NOTICE

Danger of collision!

If 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

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | CIRCULAR POCKET (Cycle 252, ISO: G252, software option 19)

6

Cycle parameters

- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

- ▶ **Q223 Circle diameter?**: Diameter of the finished pocket. Input range 0 to 99999.9999

- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999

- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999

- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **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 range: 0.1 to 1.9999; alternatively **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

- Alternative: **PREDEF**
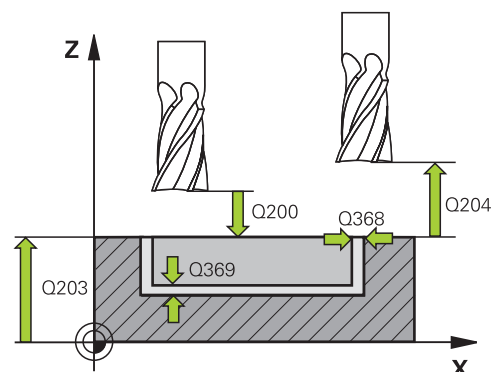
Example

8 CYCL DEF 252 CIRCULAR POCKET	
Q215=0	;MACHINING OPERATION
Q223=60	;CIRCLE DIAMETER
Q368=0.2	;ALLOWANCE FOR SIDE
Q207=500	;FEED RATE FOR MILLNG
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
Q370=1	;TOOL PATH OVERLAP
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=3	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99	

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | CIRCULAR POCKET (Cycle 252, ISO: G252, software option 19)

6

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
0: Feed rate with respect to the tool center point path
1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
3: Feed rate always refers to the cutting edge

6.4 SLOT MILLING (Cycle 253, DIN/ISO: G253), Software Option 19

Cycle run

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

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 (parameters Q368 and Q369) into account

3 The control retracts the tool to the 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 in the left slot arc

6 Then the control finishes the floor of the slot from the inside out.

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | SLOT MILLING (Cycle 253, DIN/ISO: G253), Software Option 19

6

Please note while programming:

ℹ️ With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

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.

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start!

▶ Do **not** program any incremental dimensions after this cycle

▶ Program an absolute position in all main axes after this cycle

---

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

▶ **Q218 Length of slot?** (value parallel to the reference axis of the working plane): Enter the length of the slot. Input range: 0 to 99999.9999

▶ **Q219 Width of slot?** (value parallel to the minor axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999

▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

▶ **Q374 Angle of rotation?** (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: -360.000 to 360.000

▶ **Q367 Position of slot (0/1/2/3/4)?**: Position of the slot in reference to the position of the tool when the cycle is called:
0: Tool position = slot center
1: Tool position = left end of slot
2: Tool position = center of left slot arc
3: Tool position = center of right slot arc
4: Tool position = right end of slot

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | SLOT MILLING (Cycle 253, DIN/ISO: G253), Software Option 19

6

- ▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

- ▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: -99999.9999 to 99999.9999

- ▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

- ▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999

- ▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999

Example

8 CYCL DEF 253 SLOT MILLING	
Q215=0	;MACHINING OPERATION
Q218=80	;SLOT LENGTH
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q374=+0	;ANGLE OF ROTATION
Q367=0	;SLOT POSITION
Q207=500	;FEED RATE FOR MILLNG

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **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

  ■ Alternative: **PREDEF**

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
0: Feed rate with respect to the tool center point path
1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
3: Feed rate always refers to the cutting edge

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=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE

9 L X+50 Y+50 R0 FMAX M3 M99

6.5 CIRCULAR SLOT (Cycle 254, ISO: G254, software option 19)

Cycle run

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

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 (parameters Q368 and Q369) into account

3 The control retracts the tool to the 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

Please note while programming:

> **ℹ** With an inactive tool table you must always plunge vertically (Q366=0) because you cannot define a plunging angle.
>
> Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).
>
> The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
>
> The position at the end of the cycle does not have to correspond to the position at cycle start! 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.
>
> The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
>
> 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.
>
> The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.
>
> The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start!

▶ Do not program any incremental dimensions after this cycle

▶ Program an absolute position in all main axes after this cycle

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | CIRCULAR SLOT (Cycle 254, ISO: G254, software option 19)

6

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

▶ Conduct a roughing operation beforehand
▶ Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

---

Cycle parameters

- ▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

- ▶ **Q219 Width of slot?** (value parallel to the minor axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999

- ▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- ▶ **Q375 Pitch circle diameter?**: Enter the diameter of the pitch circle. Input range: 0 to 99999.9999

- ▶ **Q367 Ref. for slot pos. (0/1/2/3)?**: Position of the slot in reference to the position of the tool when the cycle is called:
  0: 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 arc. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account
  2: Tool position = center of centerline. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account
  3: Tool position = center of right slot arc. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.

- ▶ **Q216 Center in 1st axis?** (absolute): Center of the pitch circle in the reference axis of the working plane. **Only effective if Q367 = 0**. Input range: -99999.9999 to 99999.9999

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | CIRCULAR SLOT (Cycle 254, ISO: G254, software option 19)

6

▶ **Q217 Center in 2nd axis?** (absolute): Center of the pitch circle in the minor axis of the working plane. **Only effective if Q367 = 0**. Input range: -99999.9999 to 99999.9999

▶ **Q376 Starting angle?** (absolute): Enter the polar angle of the starting point. Input range: -360.000 to 360.000

▶ **Q248 Angular length?** (incremental): Enter the angle between the starting point and the end point of the slot. Input range: 0 to 360.000

▶ **Q378 Intermediate stepping angle?** (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. Input range: -360.000 to 360.000

▶ **Q377 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range: 1 to 99999

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
+1 = Climb
-1 = Up-cut
PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: -99999.9999 to 99999.9999

▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

Example

8 CYCL DEF 254 CIRCULAR SLOT	
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q375=80	;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=0	;STEPPING ANGLE
Q377=1	;NR OF REPETITIONS

▶ **Q338 Infeed for finishing?** (incremental):
Infeed in the spindle axis per finishing cut.
Q338=0: Finishing in one infeed. Input range: 0 to
99999.9999

▶ **Q200 Set-up clearance?** (incremental): Distance
between tool tip and workpiece surface. Input
range 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute):
Coordinate of the workpiece surface. Input range
-99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental):
Coordinate in the spindle axis at which no collision
between tool and workpiece (fixtures) can occur.
Input range 0 to 99999.9999

▶ **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: reciprocal plunging. In the tool table, the
plunging angle **ANGLE** for the active tool must be
defined as not equal to 0. Otherwise, the control
generates an error message
PREDEF: The control uses the value from the
GLOBAL DEF block

▶ **Q385 Finishing feed rate?**: Traversing speed of
the tool in mm/min during side and floor finishing.
Input range: 0 to 99999,999; alternatively **FAUTO**,
FU, **FZ**

▶ **Q439 Feed rate reference (0-3)?**: Specify what
the programmed feed rate refers to:
0: Feed rate with respect to the tool center point
path
1: Feed rate with respect to the tool edge, but only
during side finishing, otherwise with respect to
the tool center path
2: Feed rate refers to the tool cutting edge during
side finishing **and** floor finishing; otherwise, it
refers to the tool center path
3: Feed rate always refers to the cutting edge

Q207=500	;FEED RATE FOR MILLNG
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=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99	

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | RECTANGULAR STUD (Cycle 256, ISO: G256, software option 19)

6

6.6 RECTANGULAR STUD (Cycle 256, ISO: G256, software option 19)

Cycle run

Use Cycle 256 to machine a rectangular stud. If a dimension of the workpiece blank is greater than the maximum possible stepover, then the control performs multiple stepovers until the finished dimension has been machined.

1 The tool moves from the cycle starting position (stud center) to the starting position for stud machining. Specify the starting position with parameter Q437. The standard setting (**Q437**=0) lies 2 mm to the right next to the stud blank.

2 If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to the set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging

3 The tool then moves 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

Please note while programming:

ℹ️ Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (position).

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.
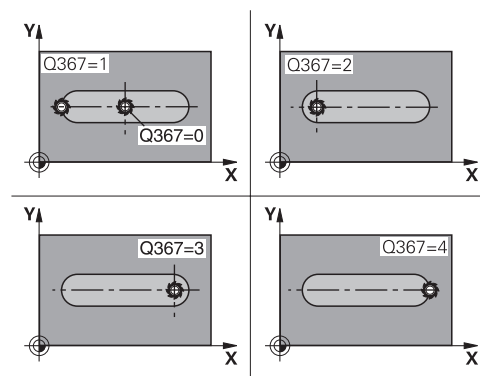

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

NOTICE

Danger of collision!

If there is not enough room for the approach movement next to the stud, there is danger of collision.

▶ Depending on the approach position Q439, leave enough room next to the stud for the approach movement
▶ Leave room next to the stud for the approach motion
▶ At least tool diameter + 2 mm
▶ At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position.

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | RECTANGULAR STUD (Cycle 256, ISO: G256, software option 19)

6

Cycle parameters

▶ **Q218 First side length?**: Stud length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q424 Workpiece blank side length 1?**: Length of the stud blank, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **First side length**. The control performs multiple lateral stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999

▶ **Q219 Second side length?**: Stud length, parallel to the minor axis of the working plane. Enter **Workpiece blank side length 2** greater than **Second side length**. The control performs multiple lateral stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999

▶ **Q425 Workpiece blank side length 2?**: Length of the stud blank, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q220 Radius / Chamfer (+/-)?**: Enter the value for the radius or chamfer form element. If you enter a positive value between 0 and +99999.9999, the control rounds every corner. The value you enter here refers to the radius. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.

▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane, is left over after machining. Input range: 0 to 99999.9999

▶ **Q224 Angle of rotation?** (absolute): Angle by which the entire machining configuration is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: -360.0000 to 360.0000

▶ **Q367 Position of stud (0/1/2/3/4)?**: Position of the stud in reference to the position of the tool when the cycle is called:
0: Tool position = stud center
1: Tool position = lower left corner
2: Tool position = lower right corner
3: Tool position = upper right corner
4: Tool position = upper left corner

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
+1 = Climb
-1 = Up-cut
PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999

▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **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 range: 0.1 to 1.9999; alternatively **PREDEF**

Example

8 CYCL DEF 256 RECTANGULAR STUD	
Q218=60	;FIRST SIDE LENGTH
Q424=74	;WORKPC. BLANK SIDE 1
Q219=40	;2ND SIDE LENGTH
Q425=60	;WORKPC. BLANK SIDE 2
Q220=5	;CORNER RADIUS
Q368=0.2	;ALLOWANCE FOR SIDE
Q224=+0	;ANGLE OF ROTATION
Q367=0	;STUD POSITION
Q207=500	;FEED RATE FOR MILLNG
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;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=+0	;FEED RATE FOR FINISHING
9 L X+50 Y+50 R0 FMAX M3 M99	

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | RECTANGULAR STUD (Cycle 256, ISO: G256, software option 19)

6

▶ **Q437 Starting position (0...4)?**: Define the approach strategy of the tool:
0: 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 should be appear on the stud surface during approach with the setting Q437=0, then choose another approach position.

▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

▶ **Q369 Finishing allowance for floor?**
(incremental): Finishing allowance for the floor.
Input range: 0 to 99999.9999

▶ **Q338 Infeed for finishing?** (incremental):
Infeed in the spindle axis per finishing cut.
Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing.
Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

6.7 CIRCULAR STUD (Cycle 257, ISO: G257, software option 19)

Cycle run

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.

1 If the tool is below the 2nd set-up clearance, the control retracts the tool 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 the 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 departs on a tangential path and then retracts in the tool axis to the 2nd set-up clearance defined in the cycle.

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | CIRCULAR STUD (Cycle 257, ISO: G257, software option 19)

6

Please note while programming:

> **ℹ** Pre-position the tool in the machining plane to the starting position (stud center) with radius compensation **R0**.
>
> The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
>
> The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
>
> At the end of the cycle, the control returns the tool to the starting position.
>
> The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative

▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

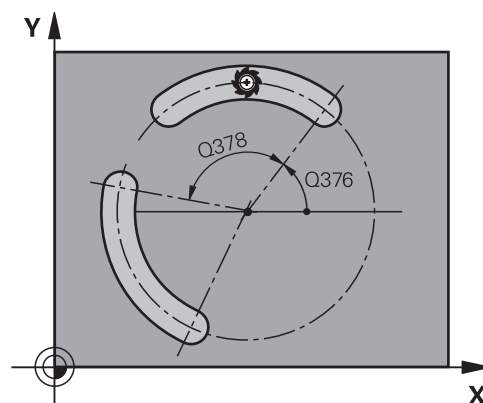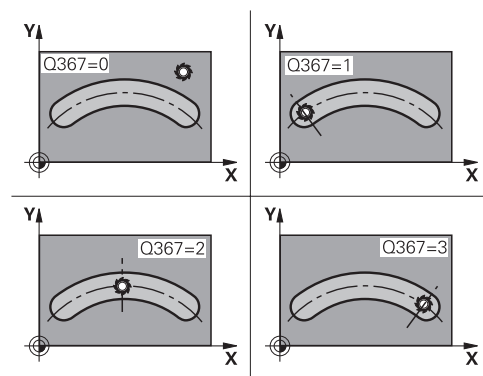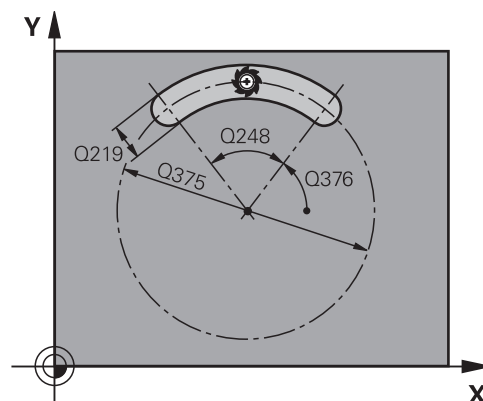### *NOTICE*

Danger of collision!

There is a danger of collision if there is insufficient room next to the stud.

▶ In this cycle, the control performs an approach movement

▶ To define the precise starting position, enter a starting angle of 0° to 360° in parameter Q376

▶ Depending on the starting angle Q376, the following amount of space must be left next to the stud: At least tool diameter +2 mm

▶ If you use the default value -1 the control automatically calculates the starting position

Cycle parameters

▶ **Q223 Finished part diameter?**: Diameter of the completely machined stud. Input range: 0 to 99999.9999

▶ **Q222 Workpiece blank diameter?**: Diameter of the 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 range: 0 to 99999.9999

▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
+1 = Climb
-1 = Up-cut
PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999

▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1,9999; alternatively **PREDEF**

▶ **Q376 Starting angle?**: Polar angle relative to the stud center from where the tool approaches the stud. Input range: 0 to 359°

▶ **Q215 Machining operation (0/1/2)?**: Define the scope of machining:
0: Roughing and finishing
1: Roughing only
2: Finishing only

▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999

▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

Example

8 CYCL DEF 257 CIRCULAR STUD	
Q223=60	;FINISHED PART DIA.
Q222=60	;WORKPIECE BLANK DIA.
Q368=0.2	;ALLOWANCE FOR SIDE
Q207=500	;FEED RATE FOR MILLNG
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;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=0	;STARTING ANGLE
Q215=+1	;MACHINING OPERATION
Q369=0	;ALLOWANCE FOR FLOOR
Q338=0	;INFEED FOR FINISHING
Q385=+500	;FINISHING FEED RATE
9 L X+50 Y+50 R0 FMAX M3 M99	

6.8 POLYGON STUD (Cycle 258, ISO: G258, software option 19)

Cycle run

With the cycle **Polygon stud** you can create an even polygon by machining the contour outside. The milling operation is carried out on a spiral path, based on the diameter of the workpiece blank.

1   If the tool is below the 2nd set-up clearance at the beginning of machining, the control retracts the tool to the 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 others, 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 the 2nd setup 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 the 2nd set-up clearance
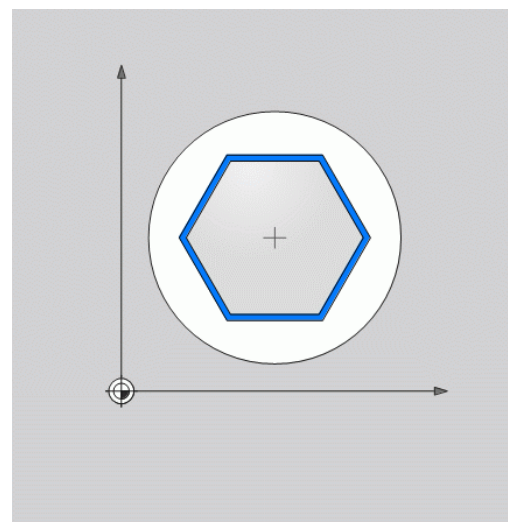
Please note while programming:

ℹ Before the start of the cycle you will have to preposition the tool on the machining plane. In order to do so, move the tool with radius compensation **R0** to the center of the stud.

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

---

NOTICE

Danger of collision!

In this cycle, the control performs an automatic approach movement. If there is not enough space, a collision might occur.

▶ Specify with Q224 which angle is used to machine the first corner of the polygon. Input range: -360° to +360°
▶ Depending on the rotary position Q224, the following amount of space must be left next to the stud: At least tool diameter + 2mm.

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.

▶ Control the traversing movements of the machine
▶ In the simulation, control the end position of the tool after the cycle
▶ After the cycle, program the absolute (not incremental) coordinates

---

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | POLYGON STUD (Cycle 258, ISO: G258, software option 19)

6

Cycle parameters

▶ **Q573 Inscr.circle/circumcircle (0/1)?**: Definition of whether the dimensioning shall reference the inscribed circle or the perimeter:
0= dimensioning refers to the inscribed circle
1= dimensioning refers to the perimeter

▶ **Q571 Reference circle diameter?**: Definition of the diameter of the reference circle. Specify in parameter Q573 whether the diameter references the inscribed circle or the perimeter. Input range: 0 to 99999.9999

▶ **Q222 Workpiece blank diameter?**: Definition of the diameter of the workpiece 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 range: 0 to 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 range: 3 to 30

▶ **Q224 Angle of rotation?**: Specify which angle is used to machine the first corner of the polygon stud. Input range: -360° to +360°
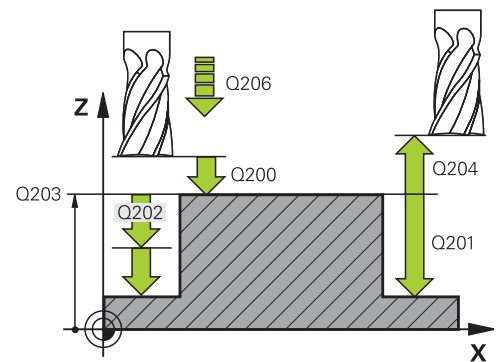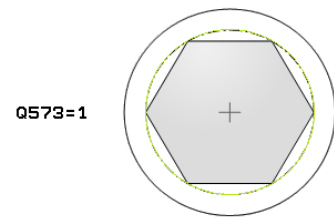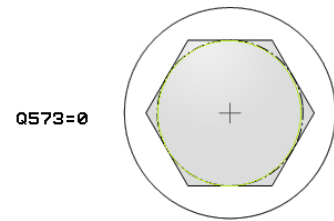
▶ **Q220 Radius / Chamfer (+/-)?**: Enter the value for the radius or chamfer form element. If you enter a positive value between 0 and +99999.9999, the control rounds every corner. The value you enter here refers to the radius. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.

▶ **Q368 Finishing allowance for side?** (incremental): 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. Input range -99999.9999 to 99999.9999

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
+1 = Climb
-1 = Up-cut
PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999

▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**

Example

8 CYCL DEF 258 POLYGON STUD	
Q573=1	;REFERENCE CIRCLE
Q571=50	;REF-CIRCLE DIAMETER
Q222=120	;WORKPIECE BLANK DIA.
Q572=10	;NUMBER OF CORNERS
Q224=40	;ANGLE OF ROTATION
Q220=2	;RADIUS / CHAMFER
Q368=0	;ALLOWANCE FOR SIDE
Q207=3000	;FEED RATE FOR MILLNG
Q351=1	;CLIMB OR UP-CUT
Q201=-18	;DEPTH
Q202=10	;PLUNGING DEPTH
Q206=150	;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
9 L X+50 Y+50 R0 FMAX M3 M99	

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | POLYGON STUD (Cycle 258, ISO: G258, software option 19)

6

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1,9999; alternatively **PREDEF**

▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

▶ **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999

▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

6.9 FACE MILLING (Cycle 233, ISO: G233, software option 19)

Cycle run

With Cycle 233, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover 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

1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane: The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.

2 The control then positions the tool at rapid traverse **FMAX** to the set-up clearance in the spindle axis

3 The tool then moves in the tool axis at the feed rate for milling Q207 to the first plunging depth calculated by the control

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | FACE MILLING (Cycle 233, ISO: G233, software option 19)

6

Strategies Q389=0 and Q389 =1

The strategies Q389=0 and Q389=1 differ in the overtravel during face milling. If Q389=0, the end point lies outside of the surface, with Q389=1, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy Q389=0 is used, the control additionally moves the tool beyond the level surface by the tool radius.

4   The control moves the tool to end point **2** at the programmed feed rate for milling

5   Then the control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side

6   The tool then returns in the opposite direction at the feed rate for milling

7   The process is repeated until the programmed surface has been completed.

8   The control then positions the tool at rapid traverse **FMAX** back to starting point **1**

9   If more than one infeed is required, the control moves the tool in the tool axis to the next plunging depth at the positioning feed rate

10 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate

11 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance
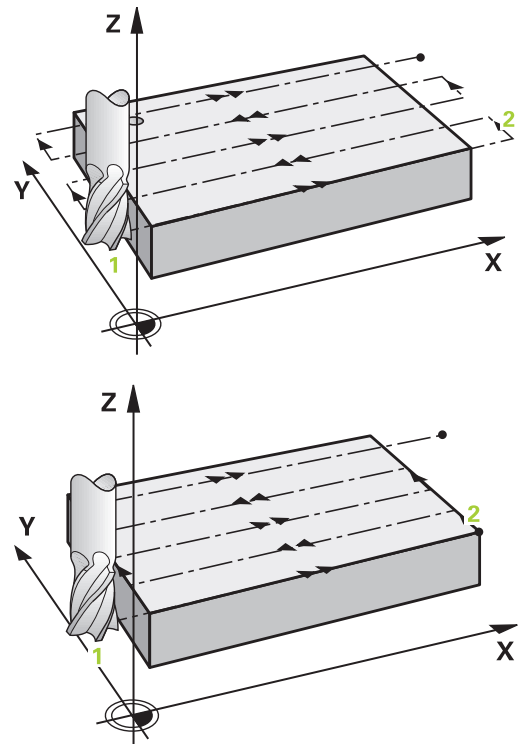
Strategies Q389=2 and Q389 =3

The strategies Q389=2 and Q389=3 differ in the overtravel during face milling. If Q389=2, the end point lies outside of the surface, with Q389=3, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy Q389=2 is used, the control additionally moves the tool beyond the level surface by the tool radius.

4 The tool subsequently moves to end point **2** at the programmed feed rate for milling

5 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves at **FMAX**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 and the set-up clearance to the side.

6 The tool then returns to the current infeed depth and moves in the direction of end point **2**

7 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**

8 If more than one infeed is required, the control moves the tool in the tool axis to the next plunging depth at the positioning feed rate

9 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate

10 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | FACE MILLING (Cycle 233, ISO: G233, software option 19)

6
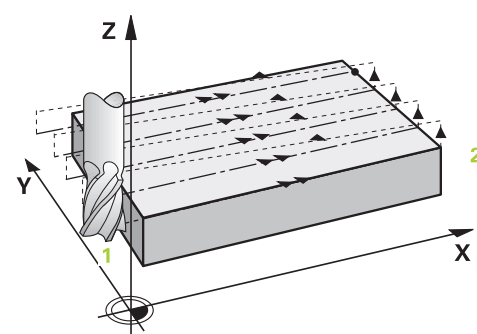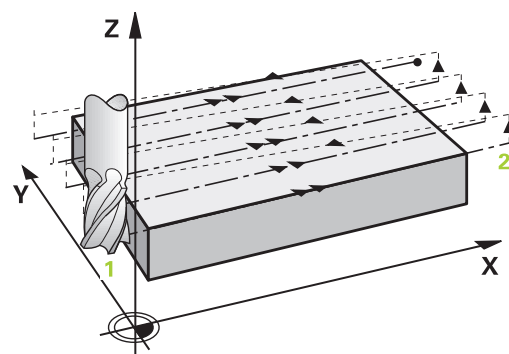
Strategy Q389=4

4   The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a tangential arc

5   The control machines the level surface at the feed rate for milling from the outside toward the inside with ever-shorter milling paths. The constant stepover results in the tool being continuously engaged

6   The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**

7   If more than one infeed is required, the control moves the tool in the tool axis to the next plunging depth at the positioning feed rate

8   The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate

9   At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**
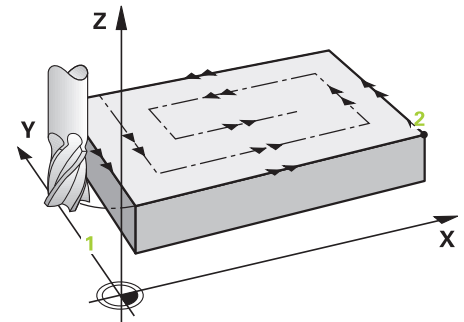
Limits

The limits enable you to set limits to the machining of the level surface so that, for example, side walls or shoulders are considered during machining. A side wall that is defined by a limit is machined to the finished dimension resulting from the starting point or the side lengths of the level surface. During roughing, the control takes the allowance for the side into account, whereas during finishing, the allowance is used for pre-positioning the tool.
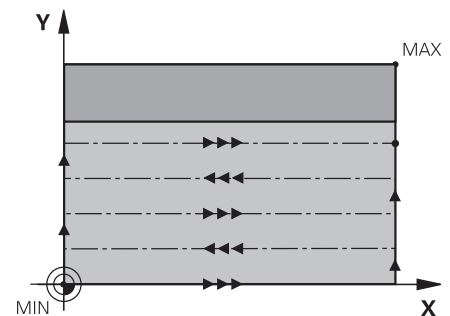
Please note while programming:

i Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Keep in mind the machining direction.

The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

Enter **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.

If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).

The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the Q202 plunging depth programmed in the cycle.

If you define **Q370** TOOL PATH OVERLAP >1, the programmed overlap factor will be taken into account right from the first machining path.

Cycle 233 monitors the entries made for the tool/cutting edge length in **LCUTS** from the tool table. If the tool or cutting edge length is not sufficient for a finishing operation, the control will subdivide the process into multiple machining steps.

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

▶ Enter depth as negative
▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | FACE MILLING (Cycle 233, ISO: G233, software option 19)

6

Cycle parameters

▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

▶ **Q389 Machining strategy (0-4)?**: Determine how the control should machine the surface:
0: Meander machining, stepover at the positioning feed rate outside the surface being machined
1: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined
2: Line-by-line machining, retraction and stepover at the positioning feed rate outside the surface to be machined
3: Line-by-line machining, retraction and stepover at positioning feed rate at the edge of the surface to be machined
4: Helical machining, uniform infeed from the outside toward the inside

▶ **Q350 Milling direction?**: Axis in the working plane that defines the machining direction:
1: Reference axis = machining direction
2: Minor axis = machining direction

▶ **Q218 First side length?** (incremental): Length of the surface to be machined in the reference axis of the working plane, referencing the starting point in the 1st axis. Input range -99999.9999 to 99999.9999

▶ **Q219 Second side length?** (incremental): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **STARTNG PNT 2ND AXIS**. Input range: -99999.9999 to 99999.9999
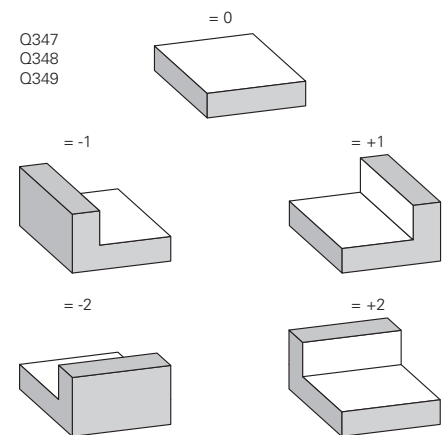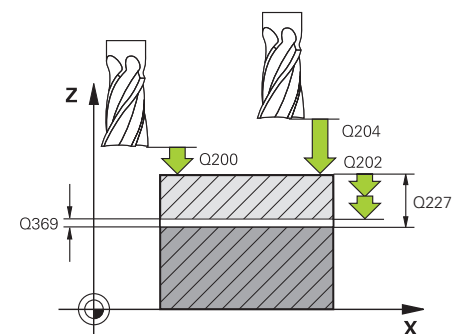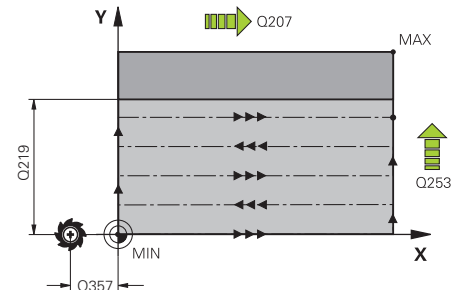
▶ **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range: -99999.9999 to 99999.9999

▶ **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis to which the surface is to be face-milled. Input range: -99999.9999 to 99999.9999

▶ **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range: 0 to 99999.9999

▶ **Q202 MAX. PLUNGING DEPTH** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

▶ **Q370 Path overlap factor?**: Maximum stepover factor k. The control calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. Input range: 0.1 to 1.9999.

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (Q389=1), the control uses the cross feed rate for milling Q207. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**

▶ **Q357 Safety clearance to the side?** (incremental) Parameter Q357 affects the following situations:
Approaching the first plunging depth: Q357 is the lateral distance between tool and workpiece
Roughing with the Q389=0-3 roughing strategies: The surface to be machined is extended in **Q350** MILLING DIRECTION by the value from Q357 if no limit has been set in that direction
Side finishing: The paths will be extended by Q357 in **Q350** MILLING DIRECTION.
Input range: 0 to 99999.9999

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

Example

8 CYCL DEF 233 FACE MILLING	
Q215=0	;MACHINING OPERATION
Q389=2	;MILLING STRATEGY
Q350=1	;MILLING DIRECTION
Q218=120	;FIRST SIDE LENGTH
Q219=80	;2ND SIDE LENGTH
Q227=0	;STARTNG PNT 3RD AXIS
Q386=-6	;END POINT 3RD AXIS
Q369=0.2	;ALLOWANCE FOR FLOOR
Q202=3	;MAX. PLUNGING DEPTH
Q370=1	;TOOL PATH OVERLAP
Q207=500	;FEED RATE FOR MILLNG
Q385=500	;FINISHING FEED RATE
Q253=750	;F PRE-POSITIONING
Q357=2	;CLEARANCE TO SIDE
Q200=2	;SET-UP CLEARANCE
Q204=50	;2ND SET-UP CLEARANCE
Q347=0	;1ST LIMIT
Q348=0	;2ND LIMIT
Q349=0	;3RD LIMIT
Q220=2	;CORNER RADIUS
Q368=0	;ALLOWANCE FOR SIDE
Q338=0	;INFEED FOR FINISHING
Q367=-1	;SURFACE POSITION (-1/0/1/2/3/4)?
9 L X+0 Y+0 R0 FMAX M3 M99	

Fixed Cycles: Pocket Milling / Stud Milling / Slot Milling | FACE MILLING (Cycle 233, ISO: G233, software option 19)

6

▸ **Q347 1st limit?**: Select the side of the workpiece where the level surface is bordered by a side wall (not possible with helical machining). Depending on the position of the side wall, the control limits machining of the level surface to the respective coordinate of the starting point or to the side length: (not possible with helical machining):
Input **0**: No limiting
Input **-1**: Limit in negative reference axis
Input **+1**: Limiting in positive reference axis
Input **-2**: Limiting in negative minor axis
Input **+2**: Limiting in positive minor axis

▸ **Q348 2nd limit?**: See parameter 1st limit Q347

▸ **Q349 3rd limit?**: See parameter 1st limit Q347

▸ **Q220 Corner radius?**: Radius of a corner at limits (Q347 to Q349). Input range: 0 to 99999.9999

▸ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

▶ **Q338 Infeed for finishing?** (incremental):
Infeed in the spindle axis per finishing cut.
Q338=0: Finishing in one infeed. Input range: 0 to
99999.9999

▶ **Q367 Surface position (-1/0/1/2/3/4)?**: Position
of the surface referencing the position of the tool
when the cycle is called:
-1: Tool position = current position
0: Tool position = stud center
1: Tool position = Lower left corner
2: Tool position = Lower right corner
3: Tool position = Upper right corner
4: Tool position = Upper left corner

6.10 Programming Examples

Example: Milling pockets, studs and slots
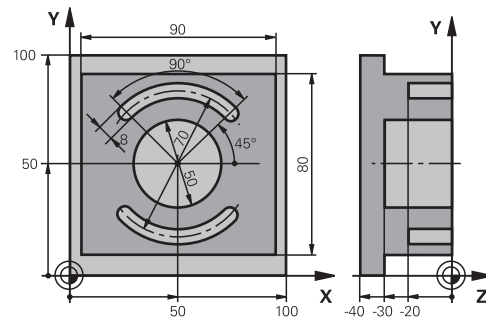
0 BEGINN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call: roughing/finishing
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD	Cycle definition: outside machining
Q218=90 ;FIRST SIDE LENGTH	
Q424=100 ;WORKPC. BLANK SIDE 1	
Q219=80 ;2ND SIDE LENGTH	
Q425=100 ;WORKPC. BLANK SIDE 2	
Q220=0 ;CORNER RADIUS	
Q368=0 ;ALLOWANCE FOR SIDE	
Q224=0 ;ANGLE OF ROTATION	
Q367=0 ;STUD POSITION	
Q207=250 ;FEED RATE FOR MILLNG	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q206=250 ;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	
6 L X+50 Y+50 R0 M3 M99	Cycle call for outside machining
7 CYCL DEF 252 CIRCULAR POCKET	Cycle definition: circular pocket
Q215=0 ;MACHINING OPERATION	
Q223=50 ;CIRCLE DIAMETER	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q207=500 ;FEED RATE FOR MILLNG	

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 L Z+250 R0 FMAX M6		Retract the tool
10 TOOL CALL 2 Z S5000		Tool call: slot milling cutter
11 CYCL DEF 254 CIRCULAR SLOT		Cycle definition: slots
Q215=0	;MACHINING OPERATION	
Q219=8	;SLOT WIDTH	
Q368=0.2	;ALLOWANCE FOR SIDE	
Q375=70	;PITCH CIRCLE DIAMETR	
Q367=0	;REF. SLOT POSITION	No pre-positioning in X/Y required
Q216=+50	;CENTER IN 1ST AXIS	
Q217=+50	;CENTER IN 2ND AXIS	
Q376=+45	;STARTING ANGLE	
Q248=90	;ANGULAR LENGTH	
Q378=180	;STEPPING ANGLE	Starting point for second slot
Q377=2	;NR OF REPETITIONS	
Q207=500	;FEED RATE FOR MILLNG	
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=1	;PLUNGE	
Q385=500	;FINISHING FEED RATE	
Q439=0	;FEED RATE REFERENCE	
12 CYCL CALL FMAX M3		Cycle call for slots
13 L Z+250 R0 FMAX M2		Retract the tool, end program

```
14 END PGM  C210 MM
```

7

Fixed Cycles:
Pattern Definitions

7.1 Fundamentals

Overview

The control provides two cycles for machining point patterns directly:

Soft key	Cycle	Page
	220 POLAR PATTERN	211
	221 CARTESIAN PATTERN	214

You can combine Cycle 220 and Cycle 221 with the following fixed cycles:

> **i** If you have to machine irregular point patterns, use **CYCL CALL PAT** (see "Point tables", Page 66) to develop point tables.
>
> More regular point patterns are available with the **pattern def** function (see "Pattern definition with PATTERN DEF", Page 59).

Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL DRILLING
Cycle 204	BACK BORING
Cycle 205	UNIVERSAL PECKING
Cycle 206	TAPPING NEW with a floating tap holder
Cycle 207	RIGID TAPPING without a floating tap holder NEW
Cycle 208	BORE MILLING
Cycle 209	TAPPING WITH CHIP BREAKING
Cycle 240	CENTERING
Cycle 251	RECTANGULAR POCKET
Cycle 252	CIRCULAR POCKET MILLING
Cycle 253	SLOT MILLING
Cycle 254	CIRCULAR SLOT (can only be combined with Cycle 221)
Cycle 256	RECTANGULAR STUD
Cycle 257	CIRCULAR STUD
Cycle 262	THREAD MILLING
Cycle 263	THREAD MILLING/COUNTERSINKING
Cycle 264	THREAD DRILLING/MILLING
Cycle 265	HELICAL THREAD DRILLING/MILLING
Cycle 267	OUTSIDE THREAD MILLING

7.2 POLAR PATTERN (Cycle 220, ISO: G220, software option 19)

Cycle run

1 The control moves the tool at rapid traverse from its current position to the starting point for the first machining operation.

Sequence:

- Move to the 2nd set-up clearance (spindle axis)
- Approach the starting point in the spindle axis.
- Move to the set-up clearance above the workpiece surface (spindle axis)

2 From this position, the control executes the last defined fixed machining cycle

3 The tool then approaches the starting point for the next machining operation on a straight lineor a circular arc. The tool stops at the set-up clearance (or the 2nd set-up clearance)

4 This process (1 to 3) is repeated until all machining operations have been executed.

Please note while programming:

ℹ Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

If you combine Cycle 220 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface and the 2nd set-up clearance that were defined in Cycle 220 or 221 will be effective. This applies within the NC program until the affected parameters are overwritten again. Example: If, in an NC program, Cycle 200 is defined with Q203=0 and you then program a Cycle 220 with Q203=-5, then the subsequent call with CYCL CALL and M99 will use Q203=-5. Cycles 220 and 221 overwrite the above-mentioned parameters of CALL-active machining cycles (if the same input parameters have been programmed in both cycles).

If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters

- ▶ **Q216 Center in 1st axis?** (absolute): Pitch circle center in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q217 Center in 2nd axis?** (absolute): Pitch circle center in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q244 Pitch circle diameter?**: Diameter of the pitch circle. Input range: 0 to 99999.9999

- ▶ **Q245 Starting angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range: -360.000 to 360.000

- ▶ **Q246 Stopping angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you specify a stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range: -360.000 to 360.000

- ▶ **Q247 Intermediate stepping angle?** (incremental): Angle between two machining operations on a pitch circle. If you enter a stepping angle of 0, the control will calculate the stepping angle from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the control will not take the stopping angle into account. The sign for the stepping angle determines the working direction (negative = clockwise). Input range: -360.000 to 360.000

- ▶ **Q241 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range: 1 to 99999

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

- ▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
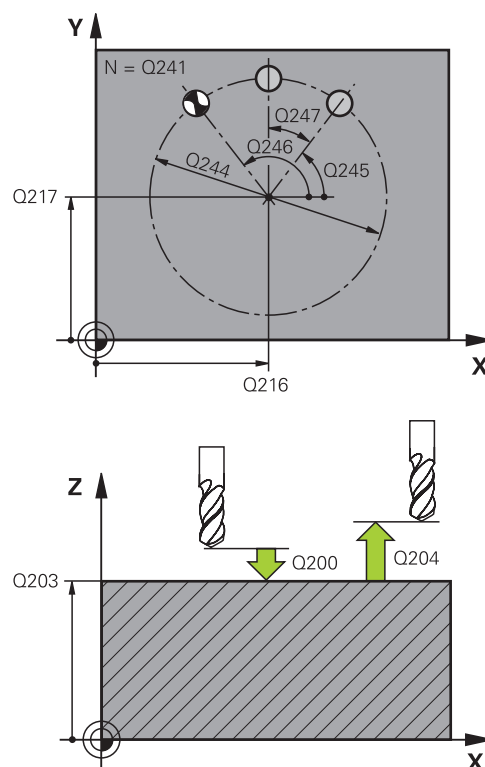
Example

53 CYCL DEF 220 POLAR PATTERN	
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q244=80	;PITCH CIRCLE DIAMETR
Q245=+0	;STARTING ANGLE
Q246=+360	;STOPPING ANGLE
Q247=+0	;STEPPING ANGLE
Q241=8	;NR OF REPETITIONS
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE
Q365=0	;TYPE OF TRAVERSE

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the tool is to move between machining operations:
0: Move to set-up clearance between machining operations
1: Move to 2nd set-up clearance between machining operations

▶ **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool will move between machining operations:
0: Move in a straight line between machining operations
1: Move in a circular arc on the pitch circle diameter between machining operations

7.3 LINEAR POINT PATTERN (Cycle 221, ISO: G221, software option 19)

Cycle run

1 The control automatically moves the tool from its current position to the starting point for the first machining operation

Sequence:

- Move to the 2nd set-up clearance (spindle axis)
- Approach the starting point in the machining plane
- Move to the set-up clearance above the workpiece surface (spindle axis)

2 From this position, the control executes the last defined fixed machining cycle

3 Then, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis. The tool stops at the set-up clearance (or the 2nd set-up clearance)

4 This procedure (steps 1 to 3) will be repeated until all machining operations from the first line have been completed. The tool is located above the last point of the first line

5 The tool subsequently moves to the last point on the second line where it carries out the machining operation

6 From this position, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis.

7 This process (6) is repeated until all machining operations in the second line have been executed.

8 The tool then moves to the starting point of the next line

9 All subsequent lines are processed in a reciprocating movement.
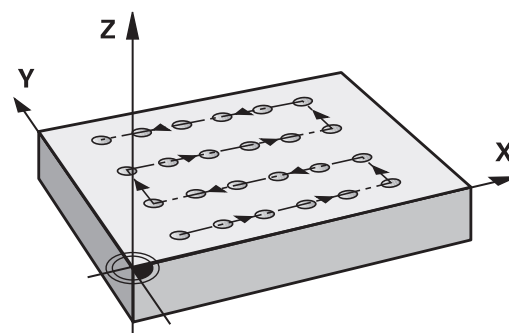
Please note while programming:

🛈 Cycle 221 is DEF active, which means that Cycle 221 automatically calls the last defined fixed cycle.

If you combine Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface, the 2nd set-up clearance, and the rotational position that were defined in Cycle 221 will be effective.

The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters
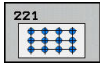
▶ **Q225 Starting point in 1st axis?** (absolute): Coordinate of the starting point in the reference axis of the working plane

▶ **Q226 Starting point in 2nd axis?** (absolute): Coordinate of the starting point in the minor axis of the working plane

▶ **Q237 Spacing in 1st axis?** (incremental): Spacing between the individual points on the line

▶ **Q238 Spacing in 2nd axis?** (incremental): Spacing between the individual lines

▶ **Q242 Number of columns?**: Number of machining operations on a line

▶ **Q243 Number of lines?**: Number of lines

▶ **Q224 Angle of rotation?** (absolute): Angle by which the entire pattern is rotated. The center of rotation is located at the starting point

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the tool is to move between machining operations:
0: Move to set-up clearance between machining operations
1: Move to 2nd set-up clearance between machining operations
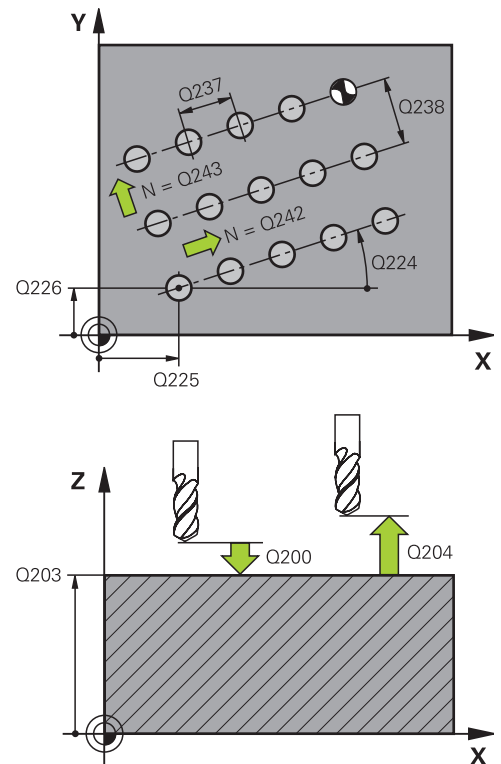
Example

54 CYCL DEF 221 CARTESIAN PATTERN	
Q225=+15	;STARTNG PNT 1ST AXIS
Q226=+15	;STARTNG PNT 2ND AXIS
Q237=+10	;SPACING IN 1ST AXIS
Q238=+8	;SPACING IN 2ND AXIS
Q242=6	;NUMBER OF COLUMNS
Q243=4	;NUMBER OF LINES
Q224=+15	;ANGLE OF ROTATION
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE

7.4 Programming Examples

Example: Polar hole patterns
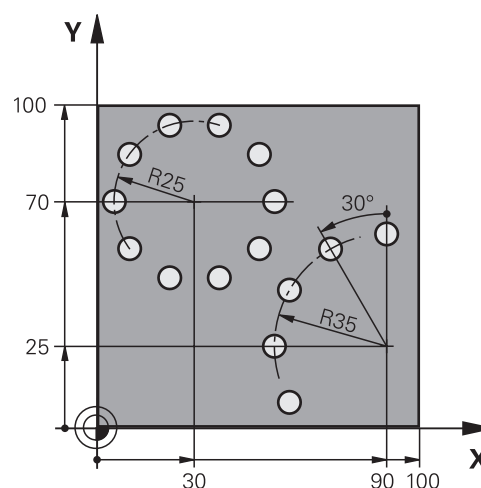
0 BEGIN PGM HOLEPAT MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call
4 L Z+250 R0 FMAX M3	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=4 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=0 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 CYCL DEF 220 POLAR PATTERN	Define cycle for polar point pattern 1, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220
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		Define cycle for polar point pattern 2, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220
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+250 R0 FMAX M2		Retract the tool, end program
9 END PGM HOLEPAT MM		

8

**Fixed Cycles:
Contour Pocket**

8.1 SL Cycles

Fundamentals

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 subcontours (subprogram numbers) that you enter in Cycle 14 CONTOUR.

i 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, you should always run a graphical program test before machining! 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

- Coordinate transformations are allowed. If they are programmed within the subcontour 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

Program structure: Machining with SL cycles

0 BEGIN PGM SL2 MM
...
12 CYCL DEF 14 CONTOUR ...
13 CYCL DEF 20 CONTOUR DATA ...
...
16 CYCL DEF 21 PILOT DRILLING ...
17 CYCL CALL
...
18 CYCL DEF 22 ROUGH-OUT ...
19 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
...

Characteristics of the fixed cycles

- 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, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

60 LBL 0
...
99 END PGM SL2 MM

Overview

Soft key	Cycle	Page
	14 CONTOUR (compulsory)	223
	20 CONTOUR DATA (compulsory)	228
	21 PILOT DRILLING (optional)	230
	22 ROUGH-OUT (compulsory)	232
	23 FLOOR FINISHING (optional)	236
	24 SIDE FINISHING (optional)	238

Enhanced cycles:

Soft key	Cycle	Page
	25 CONTOUR TRAIN	241
	270 CONTOUR TRAIN DATA	250

8.2 CONTOUR (Cycle 14, ISO: G37)

Please note while programming:

All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR.

> ℹ️ 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.
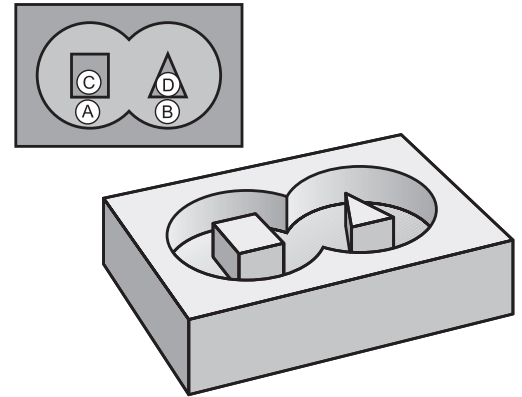
Cycle parameters

> ▶ **Label numbers for the contour**: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm each input with the ENT key. Confirm each input with the **END** key. Input of up to 12 subprogram numbers: 1 to 65 535

8.3   Superimposed contours

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.
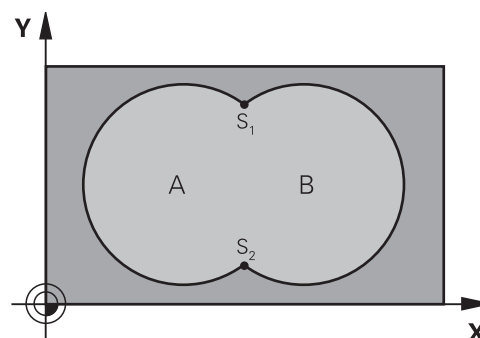
Example

12 CYCL DEF 14.0 CONTOUR
13 CYCL DEF 14.1 CONTOUR LABEL1/2/3/4

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

51 LBL 1
52 L   X+10 Y+50 RR
53 CC X+35 Y+50
54 C   X+10 Y+50 DR-
55 LBL 0

Subprogram 2: Pocket B

56 LBL 2
57 L   X+90 Y+50 RR
58 CC X+65 Y+50
59 C   X+90 Y+50 DR-
60 LBL 0

Area of inclusion

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
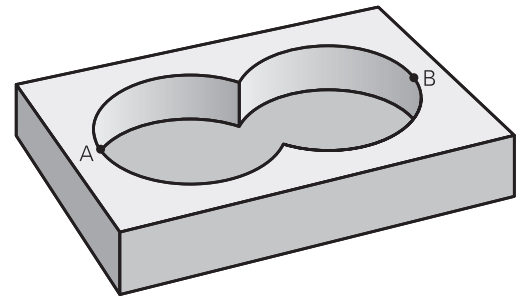
Surface A:

51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0

Surface B:

56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0

Area of exclusion

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.
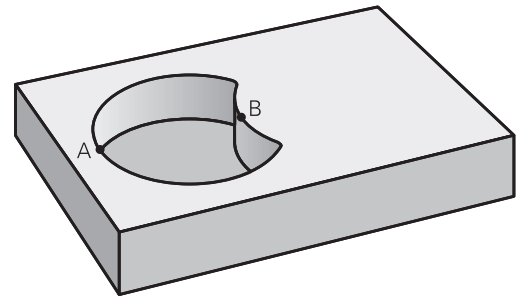
Surface A:

51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0

Surface B:

56 LBL 2
57 L X+40 Y+50 RL
58 CC X+65 Y+50
59 C X+40 Y+50 DR-
60 LBL 0

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.)

- A and B must be pockets
- A must start inside of B
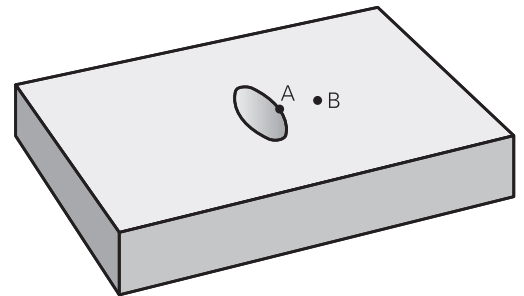
Surface A:

51 LBL 1
52 L X+60 Y+50 RR
53 CC X+35 Y+50
54 C X+60 Y+50 DR-
55 LBL 0

Surface B:

56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0

8.4 CONTOUR DATA (Cycle 20, ISO: G120, software option 19)

Please note while programming:

Use Cycle 20 to program machining data for the subprograms describing the subcontours.

> **ℹ** 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.
>
> 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.
>
> If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q20 cannot be used as program parameters.

Cycle parameters

▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range: -99999.9999 to 99999.9999

▶ **Q2 Path overlap factor?**: Q2 x tool radius = stepover factor k. Input range: -0.0001 to 1.9999

▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q4 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: -99999.9999 to 99999.9999

▶ **Q5 Workpiece surface coordinate?** (absolute): Absolute coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

▶ **Q6 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999

▶ **Q7 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: -99999.9999 to 99999.9999

▶ **Q8 Inside corner radius?**: Inside "corner" rounding radius; entered value references 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 as a separate contour element between programmed elements!** Input range: 0 to 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

You can check the machining parameters during a program interruption and overwrite them, if required.
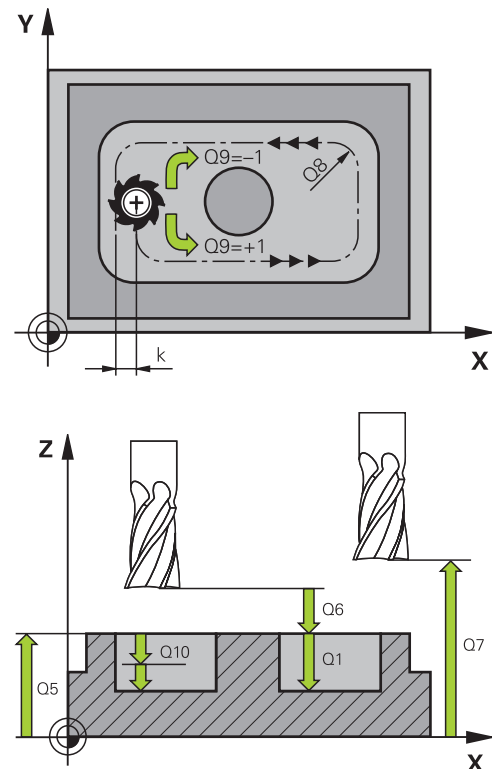
Example

57 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=+30	;SURFACE COORDINATE
Q6=2	;SET-UP CLEARANCE
Q7=+80	;CLEARANCE HEIGHT
Q8=0.5	;ROUNDING RADIUS
Q9=+1	;ROTATIONAL DIRECTION

8.5 PILOT DRILLING (Cycle 21, ISO: G121, software option 19)

Cycle run

You use Cycle 21 PILOT DRILLING if you subsequently do not use a center-cut end mill (ISO 1641) for clearing out your contour. 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 allowance for side and the 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 calling Cycle 21 you need to program two further cycles:

■ **Cycle 14 CONTOUR** or SEL CONTOUR—needed by Cycle 21 PILOT DRILLING in order to determine the drilling position in the plane

■ **Cycle 20 CONTOUR DATA**—needed by Cycle 21 PILOT DRILLING in order to determine parameters such as hole depth and set-up clearance

Cycle run:

1 The tool first positions the tool in the plane (the position results from the contour that you previously defined with Cycle 14 or SEL CONTOUR and the information on the rough-out tool)

2 The tool then moves at rapid traverse **FMAX** to the set-up clearance. (Define 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 depends on the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

ℹ 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 TNC uses the data of the tool that is currently in the spindle.

After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have set the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter to **ToolAxClearanceHeight**.

Cycle parameters

▶ **Q10 Plunging depth?** (incremental): Dimension by which the tool drills in each infeed (minus sign for negative working direction). Input range: -99999.9999 to 99999.9999

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q13 Rough-out tool number/name?** or QS13: Number or name of rough-out tool. It is possible to take over the tool via soft key directly from the tool table.
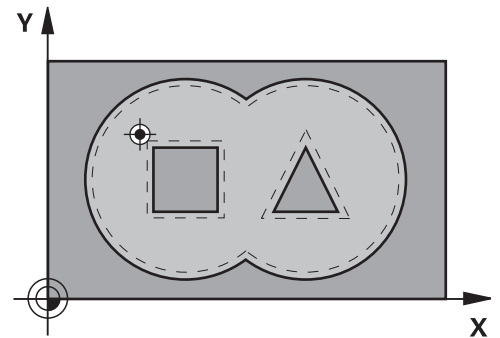
Example

58 CYCL DEF 21 PILOT DRILLING	
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q13=1	;ROUGH-OUT TOOL

8.6 ROUGHING (Cycle 22, ISO: G122, software option 19)

Cycle run

Use Cycle 22 ROUGHING to define the technology data for roughing.

Before calling Cycle 22 you need to program further cycles:

- Cycle 14 CONTOUR or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary

Cycle run

1 The control positions the tool above the cutter infeed point, taking the allowance for side into account

2 In the first plunging depth, the tool mills the contour from inside outward at the milling feed rate.

3 First the island contours (C and D in the figure at right) are rough-milled until the pocket contour (A, B) is approached.

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 depends on the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.
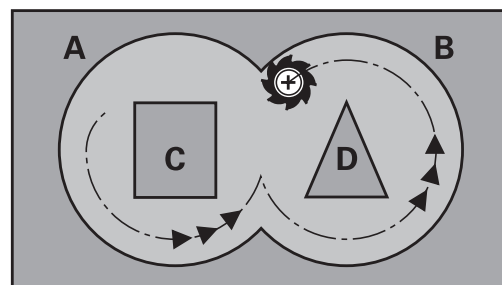
Please note while programming:

ℹ️ This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.

You define the plunging behavior of Cycle 22 with parameter Q19 and with the tool table in the **ANGLE** and **LCUTS** columns:

- 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 control plunges helically using the defined **ANGLE**

- If the reciprocation feed is defined in Cycle 22 and no **ANGLE** can be found in the tool table, the control displays an error message

- If geometrical 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**)

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.

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 of compensated circular arcs within will be reduced accordingly.

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.

- ▶ 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, no incremental traverse

Cycle parameters

▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. 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 that is 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** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered

▶ **Q19 Feed rate for reciprocation?**: Traversing speed of the tool in mm/min during reciprocating plunge-cut. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **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 Q12. Input range: 0 to 99999,9999; alternatively **FMAX, FAUTO**

Example

59 CYCL DEF 22 ROUGH-OUT	
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=750	;FEED RATE F. ROUGHNG
Q18=1	;COARSE ROUGHING TOOL
Q19=150	;FEED RATE FOR RECIP.
Q208=9999	;RETRACTION FEED RATE
Q401=80	;FEED RATE FACTOR
Q404=0	;FINE ROUGH STRATEGY

▶ **Q401 Feed rate factor in %?**: Percentage factor by 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 range: 0.0001 to 100.0000

▶ **Q404 Fine roughing strategy (0/1)?**: Specify the fine roughing behavior of the control if the radius of the fine roughing tool is greater than or equal to half the radius of the coarse roughing tool:
Q404=0:
The control moves the tool between the areas to be fine-roughed at the current depth along the contour
Q404=1:
The control retracts the tool to the set-up clearance between the areas to be fine-roughed and then moves to the starting point for the next area to be roughed-out

8.7 FLOOR FINISHING (Cycle 23, ISO: G123, software option 19)

Cycle run

With Cycle 23 FLOOR FINISHING, you can clear the finishing allowance for floor that is 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 calling Cycle 23 you need to program further cycles:

■ Cycle 14 CONTOUR or SEL CONTOUR

■ Cycle 20 CONTOUR DATA

■ Cycle 21 PILOT DRILLING, if necessary

■ Cycle 22 ROUGHING, if necessary

Cycle run

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 depends on the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

ℹ 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 of compensated circular arcs within will be reduced accordingly.

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.

▶ 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, no incremental traverse

Cycle parameters

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **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 Q12. Input range: 0 to 99999,9999; alternatively **FMAX, FAUTO**
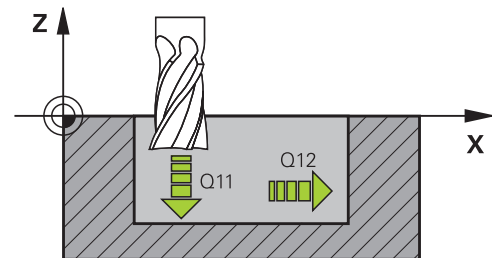
Example

60 CYCL DEF 23 FLOOR FINISHING	
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q208=9999	;RETRACTION FEED RATE

8.8 SIDE FINISHING (Cycle 24, ISO: G124, software option 19)

Cycle run

With Cycle 24 **SIDE FINISHING**, you can clear the finishing allowance for side that is programmed in Cycle 20. You can run this cycle in climb or up-cut milling.

Before calling Cycle 24 you need to program further cycles:

- Cycle 14 CONTOUR or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary
- Cycle 22 ROUGHING, if necessary

Cycle run

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 depends on the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

ℹ️ The sum of allowance for 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.

If no allowance has been defined in Cycle 20, the control issues the error message "Tool radius too large".

The allowance for side Q14 is left over after finishing. Therefore, it must be smaller than the allowance in Cycle 20.

This calculation also holds if you run Cycle 24 without having roughed out with Cycle 22; in this case, enter "0" for the radius of the rough mill.

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 limit)
- enter the finishing allowance (Q3) in Cycle 20 to be greater than the sum of the finishing allowance Q14 + radius of the tool being used

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.

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.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.

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.

- ▶ 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, no incremental traverse

Cycle parameters

▶ **Q9 Direction of rotation? cw = -1**: Machining direction:
+1: Counterclockwise rotation
-1: Clockwise rotation

▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q14 Finishing allowance for side?** (incremental): The allowance for side Q14 is left over after finishing. (This allowance must be smaller than the allowance in Cycle 20.) Input range: -99999.9999 to 99999.9999

▶ **Q438 Number/name of the rough-out tool Q438** or **QS438**: Number or name of the tool with which the control roughed out the contour pocket. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. Input range if a number is entered: -1 to +32767.9
Q438=-1: The control assumes that the tool last used was the rough-out tool (default behavior)
Q438=0: If there was no coarse-roughing, enter 0. A roughing tool with the radius 0 is assumed
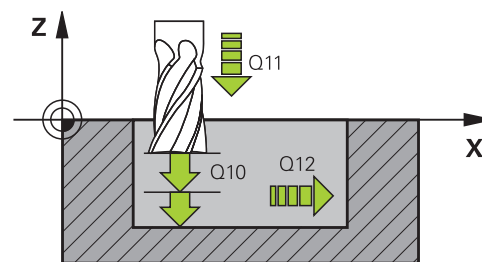
Example

61 CYCL DEF 24 SIDE FINISHING	
Q9=+1	;ROTATIONAL DIRECTION
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q14=+0	;ALLOWANCE FOR SIDE
Q438=-1	;NUMBER/NAME OF ROUGH-OUT TOOL?

8.9 CONTOUR TRAIN (Cycle 25, ISO: G125, software option 19)

Cycle run

In conjunction with Cycle 14 CONTOUR, this cycle facilitates the machining of 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. It is recommended that you 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
- The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when the contours are 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.
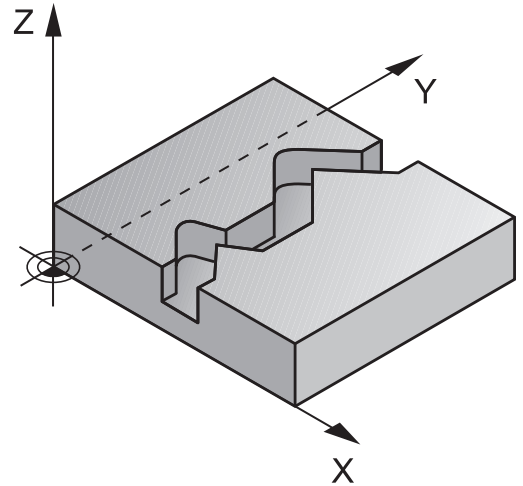
Please note while programming!

ℹ️ The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

The control takes only the first label of Cycle 14 CONTOUR into account.

If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.

Cycle 20 **CONTOUR DATA** is not required.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.

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.

▶ 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, no incremental traverse

Cycle parameters

▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999

▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q5 Workpiece surface coordinate?** (absolute): Absolute coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999

▶ **Q7 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: -99999.9999 to 99999.9999

▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q15 Climb or up-cut? up-cut = -1**:
Climb milling: Input value = +1
Up-cut milling: Input value = −1
Climb milling and up-cut milling alternately in several infeeds: Input value = 0

Example

62 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=100	;FEED RATE FOR PLNGNG
Q12=350	;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

▶ **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. 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 that is 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** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered

▶ **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 range 0.001 to 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 range: 0 to 999.9999

▶ **Q448 Path extension?** Length by which the tool path is extended at the start and end of the contour. The control always extends the tool path in parallel to the contour. Input range 0 to 99.999

8.10 THREE-D CONT. TRAIN (Cycle 276, ISO: G276, software option 19)

Cycle run

In conjunction with Cycle 14 CONTOUR and Cycle 270 CONTOUR TRAIN DATA, this cycle facilitates the machining of open and closed contours. You can also work with automatic residual material detection. This way you can subsequently complete e.g. 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.

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. in Start t.. 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. in Start t.. 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 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.
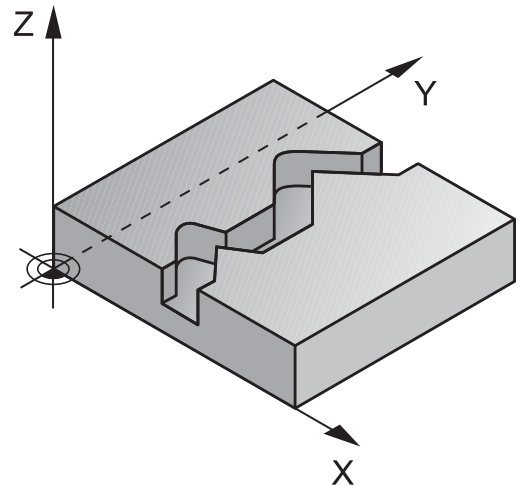
Please note while programming:

ℹ️ The first NC block in the contour subprogram must contain values in all of the three axes X, Y and Z.

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

The algebraic sign for the depth parameter determines the working direction. If you program DEPTH=0, the cycle will use the tool axis coordinates that have been specified in the contour subprogram.

If using Cycle 25 CONTOUR TRAIN, you can define only one subprogram in the CONTOUR cycle.

We recommend that you use Cycle 270 CONTOUR TRAIN DATA in conjunction with Cycle 276. Cycle 20 CONTOUR DATA, however, is not required.

If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

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 of compensated circular arcs within will be reduced accordingly.


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.

- ▶ 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, no incremental traverse

---

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

---

Cycle parameters

▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999

▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q7 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: -99999.9999 to 99999.9999

▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q15 Climb or up-cut? up-cut = -1**:
Climb milling: Input value = +1
Up-cut milling: Input value = –1
Climb milling and up-cut milling alternately in several infeeds: Input value = 0

▶ **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. 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 that is 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** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered

Example

62 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

▶ **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 range 0.001 to 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 range: 0 to 999.9999

▶ **Q448 Path extension?** Length by which the tool path is extended at the start and end of the contour. The control always extends the tool path in parallel to the contour. Input range 0 to 99.999

8.11 CONTOUR TRAIN DATA (Cycle 270, ISO: G270, software option 19)

Please note while programming:

You can use this cycle to specify various properties of Cycle 25 CONTOUR TRAIN.

> **ⓘ** 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

▶ **Q390 Type of approach/departure?**: Definition of the type of approach or departure:
Q390=1:
Approach the contour tangentially on a circular arc
Q390=2:
Approach the contour tangentially on a straight line
Q390=3:
Approach the contour at a right angle

▶ **Q391 Radius comp. (0=R0/1=RL/2=RR)?**:
Definition of the radius compensation:
Q391=0:
Machine the defined contour without radius compensation
Q391=1:
Machine the defined contour with compensation to the left
Q391=2:
Machine the defined contour with compensation to the right

▶ **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 range: 0 to 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 range: 0 to 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 is to approach the contour. Input range: 0 to 99999.9999

Example

| 62 CYCL DEF 270 CONTOUR TRAIN DATA	
Q390=1	;TYPE OF APPROACH
Q391=1	;RADIUS COMPENSATION
Q392=3	;RADIUS
Q393=+45	;CENTER ANGLE
Q394=+2	;DISTANCE

8.12 TROCHOIDAL SLOT (Cycle 275, ISO: G275, software option 19)

Cycle run

In conjunction with Cycle 14 **CONTOUR**, this cycle facilitates the complete machining of 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.

Depending on the cycle parameters you select, the following machining alternatives are available:

- Complete machining: Roughing, side finishing
- Only roughing
- Only side finishing

Roughing with closed slots

In case of a closed slot, the contour description must always start with an 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 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 with 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.

Program structure: Machining with SL cycles

0 BEGIN PGM CYC275 MM
...
12 CYCL DEF 14.0 CONTOUR
13 CYCL DEF 14.1 CONTOUR LABEL 10
14 CYCL DEF 275 TROCHOIDAL SLOT...
15 CYCL CALL M3
...
50 L Z+250 R0 FMAX M2
51 LBL 10
...
55 LBL 0
...
99 END PGM CYC275 MM

Roughing with 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 positions there perpendicular 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 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 with open slots

5   If a finishing allowance has been defined, the control finishes the slot walls, in multiple infeeds, if so 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.

Please note while programming:

ℹ The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

When using Cycle 275 TROCHOIDAL SLOT, you can 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 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 starting point of a closed slot must not be located in a contour corner.

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.

▶ 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, no incremental traverse

Cycle parameters

▶ **Q215 Machining operation (0/1/2)?**: Define machining operation:
0: Roughing and finishing
1: Only roughing
2: Only finishing
Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed

▶ **Q219 Width of slot?** (value parallel to the minor axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999

▶ **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

▶ **Q436 Feed per revolution?** (absolute): Value by which the control moves the tool in the machining direction per revolution. Input range: 0 to 99999.9999

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
+1 = Climb
-1 = Up-cut
PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: -99999.9999 to 99999.9999

▶ **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

▶ **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
0 = vertical plunging. The control plunges the tool 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
Alternatively: **PREDEF**

Example

8 CYCL DEF 275 TROCHOIDAL SLOT	
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q436=2	;INFEED PER REV.
Q207=500	;FEED RATE FOR MILLNG
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;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
9 CYCL CALL FMAX M3	

▶ **Q369 Finishing allowance for floor?**
(incremental): Finishing allowance for the floor.
Input range: 0 to 99999.9999

▶ **Q439 Feed rate reference (0-3)?**: Specify what
the programmed feed rate refers to:
0: Feed rate with respect to the tool center point
path
1: Feed rate with respect to the tool edge, but only
during side finishing, otherwise with respect to
the tool center path
2: Feed rate refers to the tool cutting edge during
side finishing **and** floor finishing; otherwise, it
refers to the tool center path
3: Feed rate always refers to the cutting edge

8.13 Programming Examples

Example: Roughing-out and fine-roughing a pocket
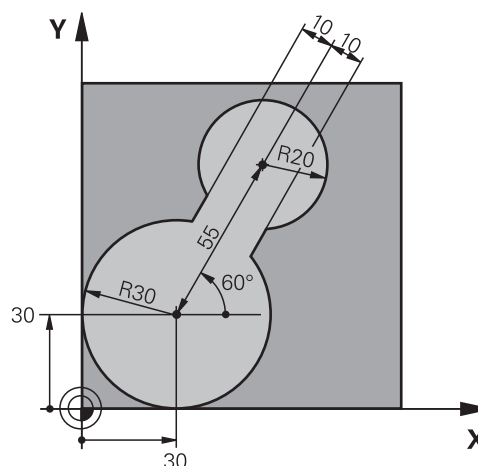
0 BEGIN PGM C20 MM	
1 BLK FORM 0.1 Z X-10 Y-10 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	Workpiece blank definition
3 TOOL CALL 1 Z S2500	Tool call: coarse roughing tool, diameter 30
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;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=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 22 ROUGH-OUT	Cycle definition: coarse roughing
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=30000 ;RETRACTION FEED RATE	
9 CYCL CALL M3	Cycle call: coarse roughing
10 L Z+250 R0 FMAX M6	Retract the tool

11 TOOL CALL 2 Z S3000	Tool call: fine roughing tool, diameter 15
12 CYCL DEF 22 ROUGH-OUT	Cycle definition: fine roughing
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE F. ROUGHNG	
Q18=1 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q208=30000 ;RETRACTION FEED RATE	
13 CYCL CALL M3	Cycle call: fine roughing
14 L Z+250 R0 FMAX M2	Retract the tool, end program
15 LBL 1	Contour subprogram
16 L X+0 Y+30 RR	
17 FC DR- R30 CCX+30 CCY+30	
18 FL AN+60 PDX+30 PDY+30 D10	
19 FSELECT 3	
20 FPOL X+30 Y+30	
21 FC DR- R20 CCPR+55 CCPA+60	
22 FSELECT 2	
23 FL AN-120 PDX+30 PDY+30 D10	
24 FSELECT 3	
25 FC X+0 DR- R30 CCX+30 CCY+30	
26 FSELECT 2	
27 LBL 0	
28 END PGM C20 MM	

Example: Pilot drilling, roughing-out and finishing overlapping contours
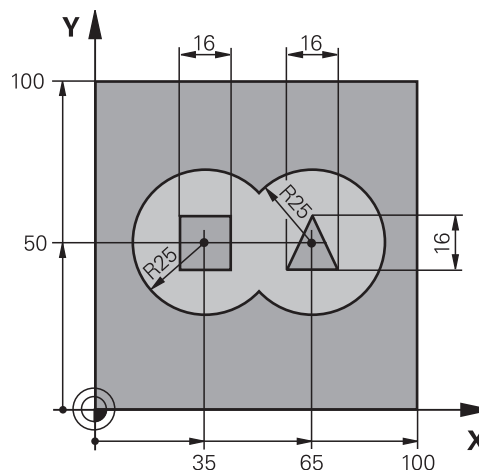
0 BEGIN PGM C21 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: drill, diameter 12
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1/2/3/4	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
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	Cycle definition: pilot drilling
Q10=5 ;PLUNGING DEPTH	
Q11=250 ;FEED RATE FOR PLNGNG	
Q13=2 ;ROUGH-OUT TOOL	
9 CYCL CALL M3	Cycle call: pilot drilling
10 L +250 R0 FMAX M6	Retract the tool
11 TOOL CALL 2 Z S3000	Tool call: roughing/finishing, diameter 12
12 CYCL DEF 22 ROUGH-OUT	Cycle definition: 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=30000	;RETRACTION FEED RATE	
13 CYCL CALL M3		Cycle call: rough-out
14 CYCL DEF 23 FLOOR FINISHING		Cycle definition: floor finishing
Q11=100	;FEED RATE FOR PLNGNG	
Q12=200	;FEED RATE F. ROUGHNG	
Q208=30000	;RETRACTION FEED RATE	
15 CYCL CALL		Cycle call: floor finishing
16 CYCL DEF 24 SIDE FINISHING		Cycle definition: 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	
17 CYCL CALL		Cycle call: side finishing
18 L Z+250 R0 FMAX M2		Retract the tool, end program
19 LBL 1		Contour subprogram 1: left pocket
20 CC X+35 Y+50		
21 L X+10 Y+50 RR		
22 C X+10 DR-		
23 LBL 0		
24 LBL 2		Contour subprogram 2: right pocket
25 CC X+65 Y+50		
26 L X+90 Y+50 RR		
27 C X+90 DR-		
28 LBL 0		
29 LBL 3		Contour subprogram 3: left square island
30 L X+27 Y+50 RL		
31 L Y+58		
32 L X+43		
33 L Y+42		
34 L X+27		
35 LBL 0		
36 LBL 4		Contour subprogram 4: right triangular island
37 L X+65 Y+42 RL		
38 L X+57		
39 L X+65 Y+58		
40 L X+73 Y+42		
41 LBL 0		
42 END PGM C21 MM		

Example: Contour train
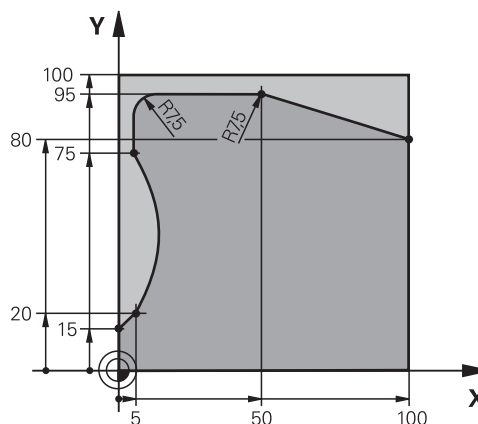
0 BEGIN PGM C25 MM		
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition	
2 BLK FORM 0.2 X+100 Y+100 Z+0		
3 TOOL CALL 1 Z S2000	Tool call: diameter 20	
4 L Z+250 R0 FMAX	Retract the tool	
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram	
6 CYCL DEF 14.1 CONTOUR LABEL 1		
7 CYCL DEF 25 CONTOUR TRAIN	Define machining parameters	
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		
Q466= 0.01 ;RESIDUAL MATERIAL		
Q447=+10 ;CONNECTION DISTANCE		
Q448=+2 ;PATH EXTENSION		
8 CYCL CALL M3	Cycle call	
9 L Z+250 R0 FMAX M2	Retract the tool, end program	
10 LBL 1	Contour subprogram	
11 L X+0 Y+15 RL		
12 L X+5 Y+20		
13 CT X+5 Y+75		
14 L Y+95		
15 RND R7.5		
16 L X+50		
17 RND R7.5		

18 L X+100 Y+80	
19 LBL 0	
20 END PGM C25 MM	

9

Fixed Cycles:
Cylindrical Surface

9.1 Fundamentals

Overview of cylindrical surface cycles

Soft key	Cycle	Page
	27 CYLINDER SURFACE	267
	28 CYLINDER SURFACE Slot milling	270
	29 CYLINDER SURFACE Ridge milling	274
	39 CYLINDER SURFACE Contour	277

9.2 CYLINDER SURFACE (Cycle 27, ISO: G127, software option 1)

Cycle run

This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle 28 if you want to mill guideways on the cylinder.

The contour is described in a subprogram identified in 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 dimensions in the rotary axis (x coordinates) can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition using Q17.

1 The control positions the tool above the cutter infeed point, taking the 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 the 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.
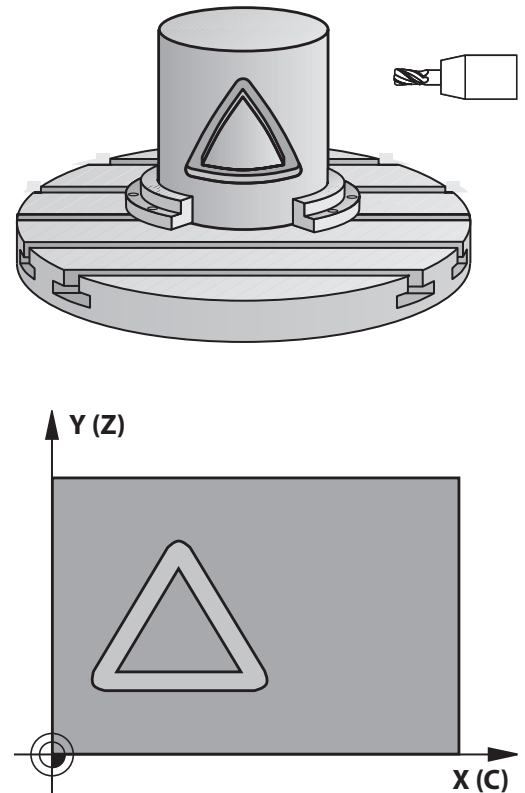
Please note while programming:

⚙ Refer to your machine manual.

The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.

ℹ In the first NC block of the contour program, always program both cylinder surface coordinates.

The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.

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 requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

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 set-up clearance must be greater than the tool radius.

The machining time can increase if the contour consists of many non-tangential contour elements.

If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

- ▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999

- ▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the plane of the developed cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range: -99999.9999 to 99999.9999

- ▶ **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999

- ▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

- ▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999

- ▶ **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches

Example

63 CYCL DEF 27 CYLINDER SURFACE	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION

9.3 CYLINDER SURFACE Slot milling (Cycle 28, ISO: G128, software option 1)

Cycle run

With this cycle you can program a guide notch 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.

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 have 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.
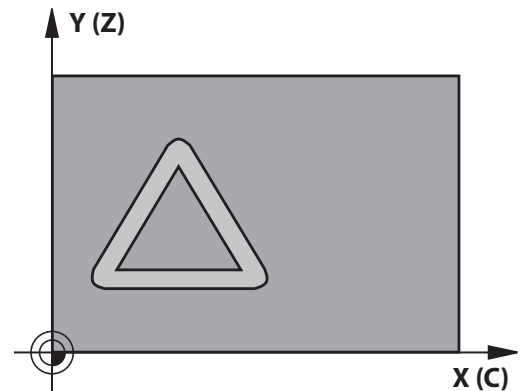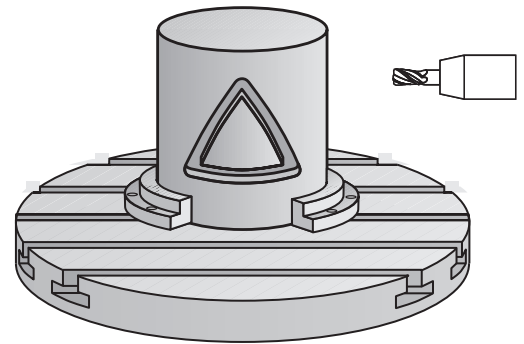
Please note while programming:

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.

Define the approaching behavior in **ConfigDatum**, **CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004)

- CircleTangential:
  Tangential approach and departure
- LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

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.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

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 set-up clearance must be greater than the tool radius.

The machining time can increase if the contour consists of many non-tangential contour elements.

If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

▶ By setting the **displaySpindleErr** 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.

▶ The function needs to be adapted by your machine manufacturer.

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.

▶ Control the traversing movements of the machine

▶ In the simulation, control the end position of the tool after the cycle

▶ After the cycle, program the absolute (not incremental) coordinates

Cycle parameters

- ▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999

- ▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. Input range: -99999.9999 to 99999.9999

- ▶ **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999

- ▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

- ▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

- ▶ **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999

- ▶ **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches

- ▶ **Q20 Slot width?**: Width of the slot to be machined. Input range: -99999.9999 to 99999.9999

- ▶ **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. Input range for tolerance: 0.0001 to 9.9999
  Recommendation: Use a tolerance of 0.02 mm.
  Function inactive: Enter 0 (default setting).

Example

63 CYCL DEF 28 CYLINDER SURFACE	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION
Q20=12	;SLOT WIDTH
Q21=0	;TOLERANCE

9.4 CYLINDER SURFACE Ridge milling (Cycle 29, ISO: G129, software option 1)

Cycle run

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.

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 stud 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.

Please note while programming:

⚙ 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.

ℹ 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.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

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.

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.

By setting parameter **CfgGeoCycle** (no. 201000), **displaySpindleErr** (no. 201002) to on/off, you can define whether the control will display an error message (on) or not (off), if the spindle is not active while the cycle is being called. The function needs to be adapted by your machine manufacturer.

Cycle parameters

▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999

▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. Input range: -99999.9999 to 99999.9999

▶ **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999

▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999

▶ **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches

▶ **Q20 Ridge width?**: Width of the ridge to be machined. Input range: -99999.9999 to 99999.9999

Example

63 CYCL DEF 29 CYL SURFACE RIDGE	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION
Q20=12	;RIDGE WIDTH

9.5 CYLINDER SURFACE CONTOUR (Cycle 39, ISO: G139, software option 1)

Cycle run

This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the developed 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.

The contour is described in a subprogram identified in 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 actual contour to be machined.

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 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 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 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.

Please note while programming:

⚙ 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.

ℹ 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.

Ensure that the tool has enough space laterally for contour approach and departure.

The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

The set-up clearance must be greater than the tool radius.

The machining time can increase if the contour consists of many non-tangential contour elements.

If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Define the approaching behavior in **ConfigDatum**, **CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004)

- CircleTangential:
  Tangential approach and departure
- LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

▶ By setting the **displaySpindleErr** 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.

▶ The function needs to be adapted by your machine manufacturer.

Cycle parameters

▶ **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999

▶ **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the plane of the developed cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range: -99999.9999 to 99999.9999

▶ **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999

▶ **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999

▶ **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999

▶ **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches

Example

63 CYCL DEF 39 CYL. SURFACE CONTOUR	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION

9.6 Programming Examples

Example: Cylinder surface with Cycle 27

> ℹ️
> - Machine with B head and C table
> - Cylinder centered on rotary table
> - Preset is on the underside, in the center of the rotary table
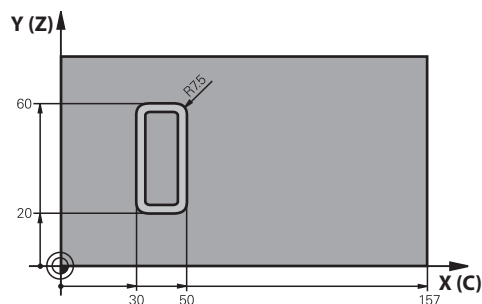
0 BEGIN PGM C27 MM	
1 TOOL CALL 1 Z S2000	Tool call, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y0 R0 FMAX	Pre-position the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MBMAX FMAX	Positioning
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 27 CYLINDER SURFACE	Define machining parameters
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 M13 M99	Pre-position rotary table, spindle ON, call the cycle
9 L Z+250 R0 FMAX	Retract the tool
10 PLANE RESET TURN FMAX	Tilt back, cancel the PLANE function
11 M2	End of program
12 LBL 1	Contour subprogram
13 L X+40 Y+20 RL	Data for the rotary axis are entered in mm (Q17=1)
14 L X+50	
15 RND R7.5	
16 L Y+60	
17 RN 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 C27 MM	

Example: Cylinder surface with Cycle 28

i ■ Cylinder centered on rotary table
■ Machine with B head and C table
■ Preset is at the center of the rotary table
■ Description of the center path in the contour subprogram
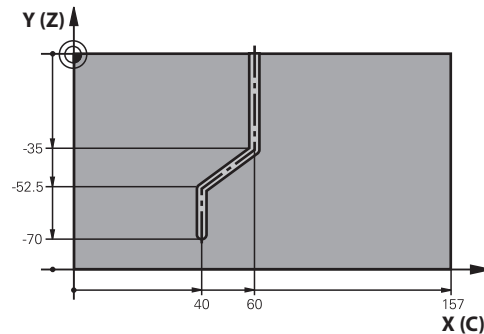
Code	Comment
0 BEGIN PGM C28 MM	
1 TOOL CALL 1 Z S2000	Tool call, tool Z axis, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y+0 R0 FMAX	Pre-position the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN FMAX	Tilting
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 28 CYLINDER SURFACE	Define machining parameters
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	Remachining active
8 L C+0 R0 FMAX M3 M99	Pre-position rotary table, spindle ON, call the cycle
9 L Z+250 R0 FMAX	Retract the tool
10 PLANE RESET TURN FMAX	Tilt back, cancel the PLANE function
11 M2	End of program
12 LBL 1	Contour subprogram, description of the center path
13 L X+60 Y+0 RL	Data for the rotary axis are entered in mm (Q17=1)
14 L Y-35	
15 L X+40 Y-52.5	
16 L Y-70	
17 LBL 0	
18 END PGM C28 MM	

10

Fixed Cycles:
Contour Pocket
with Contour
Formula

10.1 SL cycles with complex contour formula

Fundamentals

Using SL cycles and the complex contour formula, you can combine subcontours (pockets or islands) to form 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.

> **i** 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** 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.
>
> In its present form, the "SL cycles with contour formula" function requires input from several areas in the control user interface. This function will serve as a basis for further development.
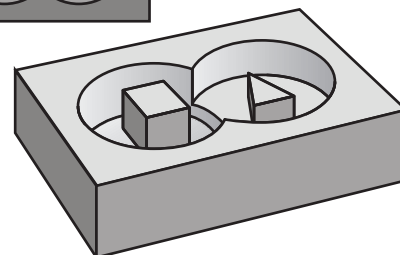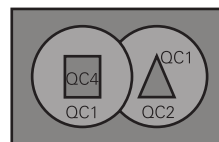
Program structure: Machining with SL cycles and complex contour formula

0 BEGIN PGM CONTOUR MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 CONTOUR DATA ...
8 CYCL DEF 22 ROUGHING ...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING ...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING ...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTOUR MM

Properties of the subcontours

- The control assumes that each contour is a pocket. Do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour 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.
- You can define subcontours with various depths as needed

Characteristics of the fixed cycles

- 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 data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Program structure: Calculation of the subcontours with contour formula

0 BEGIN PGM MODEL MM
1 DECLARE CONTOUR QC1 = "CIRCLE1"
2 DECLARE CONTOUR QC2 = "CIRCLEXY" DEPTH15
3 DECLARE CONTOUR QC3 = "TRIANGLE" DEPTH10
4 DECLARE CONTOUR QC4 = "SQUARE" DEPTH5
5 QC10 = (QC1 | QC3 | QC4) \ QC2
6 END PGM MODEL MM

0 BEGIN PGM CIRCLE1 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM CIRCLE1 MM

0 BEGIN PGM CIRCLE31XY MM
...
...

Selecting an NC program with contour definitions

With the **SEL CONTOUR** function, you select an NC program with contour NC, from which the control extracts the contour descriptions:

SPEC FCT	▶ Show the soft-key row with special functions
CONTOUR + POINT MACHINING	▶ Menu for functions: Press the soft key for contour and point machining
SEL CONTOUR	▶ Press the **SEL CONTOUR** soft key.
	▶ Enter the full name of the NC program with the contour definitions. Confirm your input with the **END** key

> ℹ️ Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR** is no longer necessary if you use **SEL CONTOUR**.

Defining contour descriptions

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 (FCL 2 function):

SPEC FCT	▶ Show the soft-key row with special functions
CONTOUR + POINT MACHINING	▶ Menu for functions: Press the soft key for contour and point machining
DECLARE CONTOUR	▶ Press the **DECLARE CONTOUR** soft key.
	▶ Enter the number for the contour designator **QC**, and confirm with the **ENT** key
	▶ Enter the full name of the NC program with the contour descriptions and confirm with the **END** key, or if desired,
	▶ Define a separate depth for the selected contour

> ℹ️ With the entered contour designators **QC** you can include the various contours in the contour formula.
>
> If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).

Entering a complex contour formula

You can use soft keys to interlink various contours in a mathematical formula.

| SPEC FCT | ▶ Show the soft-key row with special functions |

| CONTOUR + POINT MACHINING | ▶ Menu for functions: Press the soft key for contour and point machining |

| CONTOUR FORMULA | ▶ Press the **CONTOUR FORMULA** soft key. The control then displays the following soft keys: |

Soft key	Mathematical function
	Intersected with e.g. **QC10= QC1& QC5**
	Joined with e.g. **QC25= QC7| QC18**
	joined with, but without intersection e.g. **QC12 = QC5 ^ QC25**
	without e.g. **QC25 = QC1 \ QC2**
(	**Open parenthesis** e.g. **QC12 = QC1 * (QC2 + QC3)**
)	**Close parenthesis** e.g. **QC12 = QC1 * (QC2 + QC3)**
	Define single contour z. B. **QC12 = QC1**

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.
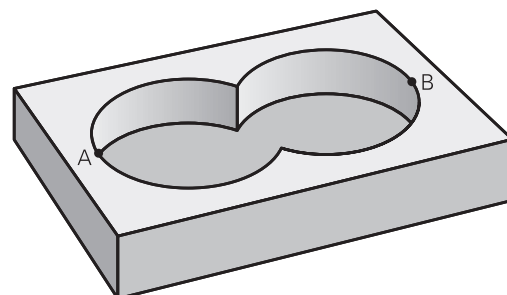
Subprograms: overlapping pockets

> **i** 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_A 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_A MM

Contour description program 2: pocket B

0 BEGIN PGM POCKET_B MM
1 L X+90 Y+50 R0
2 CC X+65 Y+50
3 C X+90 Y+50 DR-
4 END PGM POCKET_A MM

Area of inclusion

Both areas A and B are to be machined, including the overlapping area:

- 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 "joined with" function.
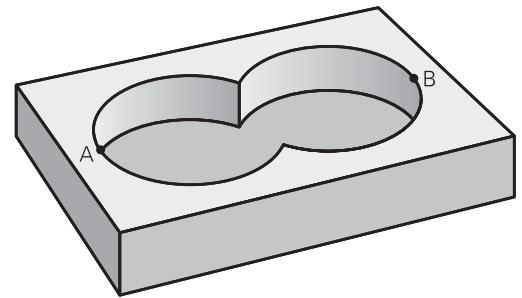
Contour definition program:

50 …
51 …
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 | QC2
55 …
56 …

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.
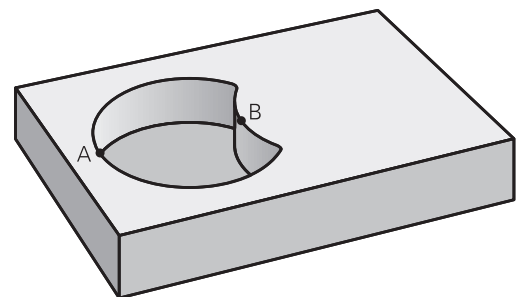
Contour definition program:

50 …
51 …
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 \ QC2
55 …
56 …

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.
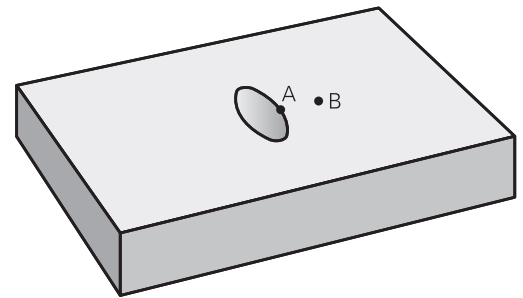
Contour definition program:

50 …
51 …
52 DECLARE CONTOUR QC1 = "POCKET_A.H"
53 DECLARE CONTOUR QC2 = "POCKET_B.H"
54 QC10 = QC1 & QC2
55 …
56 …

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", Page 222).

Example: Roughing and finishing superimposed contours with the contour formula
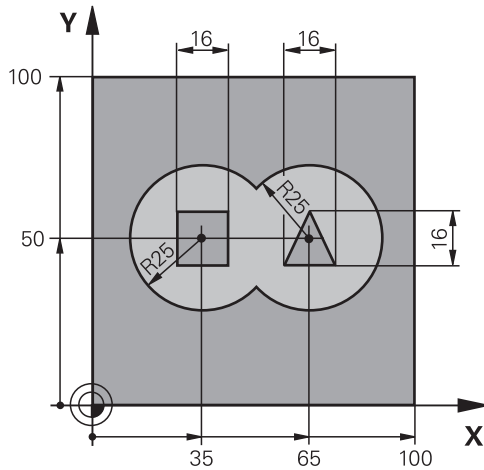
0 BEGIN PGM CONTOUR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: roughing cutter
4 L Z+250 R0 FMAX	Retract the tool
5 SEL CONTOUR "MODEL"	Define contour definition program
6 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
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	

7 CYCL DEF 22 ROUGH-OUT		Cycle definition: 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.	
Q401=100	;FEED RATE FACTOR	
Q404=0	;FINE ROUGH STRATEGY	
8 CYCL CALL M3		Cycle call: rough-out
9 TOOL CALL 2 Z S5000		Tool call: finishing cutter
10 CYCL DEF 23 FLOOR FINISHING		Cycle definition: floor finishing
Q11=100	;FEED RATE FOR PLNGNG	
Q12=200	;FEED RATE F. ROUGHNG	
11 CYCL CALL M3		Cycle call: floor finishing
12 CYCL DEF 24 SIDE FINISHING		Cycle definition: 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	
13 CYCL CALL M3		Cycle call: side finishing
14 L Z+250 R0 FMAX M2		Retract the tool, end program
15 END PGM KONTUR MM		

Contour definition program with contour formula:

0 BEGIN PGM MODEL MM	Contour definition program
1 DECLARE CONTOUR QC1 = "CIRCLE1"	Definition of the contour label for the "CIRCLE1" NC program
2 FN 0: Q1 =+35	Assignment of values for parameters used in PGM "CIRCLE31XY"
3 FN 0: Q2 =+50	
4 FN 0: Q3 =+25	
5 DECLARE CONTOUR QC2 = "CIRCLE31XY"	Definition of the contour label for the "CIRCLE3XY" NC program
6 DECLARE CONTOUR QC3 = "TRIANGLE"	Definition of the contour label for the "TRIANGLE" NC program
7 DECLARE CONTOUR QC4 = "SQUARE"	Definition of the contour label for the "SQUARE" NC program
8 QC10 = (QC 1 | QC 2) \ QC 3 \ QC 4	Contour formula
9 END PGM MODEL MM	

Contour description programs:

0 BEGIN PGM CIRCLE1 MM	Contour description program: circle at right
1 CC X+65 Y+50	
2 L PR+25 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE1 MM	

0 BEGIN PGM CIRCLE31XY MM	Contour description program: circle at left
1 CC X+Q1 Y+Q2	
2 LP PR+Q3 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE31XY MM	

0 BEGIN PGM TRIANGLE MM	Contour description program: triangle at right
1 L X+73 Y+42 R0	
2 L X+65 Y+58	
3 L X+58 Y+42	
4 L X+73	
5 END PGM TRIANGLE MM	

0 BEGIN PGM SQUARE MM	Contour description program: square at left
1 L X+27 Y+58 R0	
2 L X+43	
3 L Y+42	
4 L X+27	
5 L Y+58	
6 END PGM SQUARE MM	

10.2 SL cycles with simple contour formula

Fundamentals

Using SL cycles and the simple contour formula, you can easily combine up to nine subcontours (pockets or islands) to form a particular contour. 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.

> **i** 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** elements.

Program structure: Machining with SL cycles and complex contour formula

0 BEGIN PGM CONTDEF MM
…
5 CONTOUR DEF P1= "POCK1.H" I2 = "ISLE2.H" DEPTH5 I3 "ISLE3.H" DEPTH7.5
6 CYCL DEF 20 CONTOUR DATA …
8 CYCL DEF 22 ROUGHING …
9 CYCL CALL
…
12 CYCL DEF 23 FLOOR FINISHING …
13 CYCL CALL
…
16 CYCL DEF 24 SIDE FINISHING …
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTDEF MM

Properties of the subcontours

- Do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour 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.

Characteristics of the fixed cycles

- 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 data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Entering a simple contour formula

You can use soft keys to interlink various contours in a mathematical formula.

SPEC FCT	▶ Show the soft-key row with special functions
CONTOUR + POINT MACHINING	▶ Menu for functions: Press the soft key for contour and point machining
CONTOUR DEF	▶ Press the **CONTOUR DEF** soft key. The control opens the dialog for entering the contour formula.
	▶ Enter the name of the first subcontour. The first subcontour must always be the deepest pocket. Confirm with the **ENT** key.
ISLAND	▶ Specify via soft key whether the next subcontour is a pocket or an island. Confirm with the **ENT** key.
	▶ Enter the name of the second subcontour. Confirm with the **ENT** key
	▶ If needed, enter the depth of the second subcontour. Confirm with the **ENT** key.
	▶ Carry on with the dialog as described above until you have entered all subcontours.

> **i** Always start the list of subcontours 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 the depth is entered as 0, then for pockets the depth defined in the Cycle 20 is effective. Islands then rise up to the workpiece top surface!

Contour machining with SL Cycles

> **i** The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", Page 222).

11

Cycles: Coordinate Transformations

11.1   Fundamentals

Overview

Once a contour has been programmed, the control can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The control provides the following functions for coordinate transformations:

Soft key	Cycle	Page
	7 DATUM SHIFT For shifting contours directly within the NC program or from datum tables	299
	247 PRESETTING Presetting during the program run	305
	8 MIRRORING Mirroring contours	306
	10 ROTATION Rotating contours in the working plane	308
	11 SCALING FACTOR Increasing or reducing the size of contours	310
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	311
	19 WORKING PLANE Machining in a tilted coordinate system on machines with swivel heads and/or rotary tables	313

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

11.2 DATUM SHIFT (Cycle 7, ISO: G54)

Effect

A datum shift allows machining operations to be repeated at various locations on the workpiece.

After the definition of a datum shift cycle, all coordinate data will reference the new datum. The control displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

Resetting

- To shift the datum back to the coordinates X=0, Y=0 etc., program another cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.
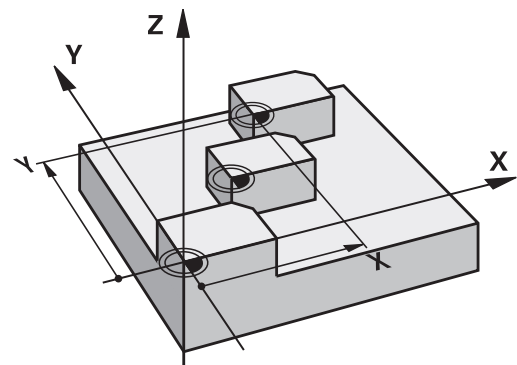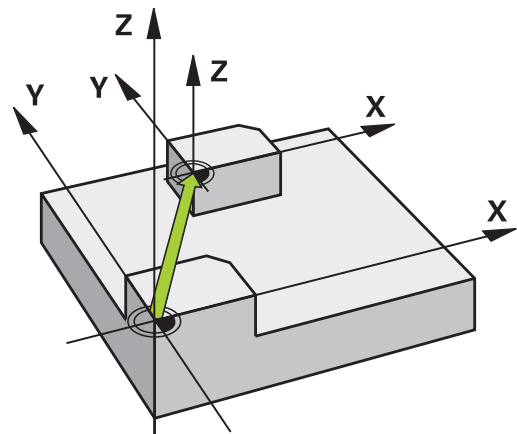
Cycle parameters
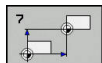
▶ **Displacement**: Enter the coordinates of the new datum. Absolute values reference the workpiece datum defined by presetting. Incremental values always reference the datum which was last valid —this can be a datum which has already been shifted. Input range: Up to six NC axes, each from –99999.9999 to 99999.9999

Example

13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 7.3 Z-5

Please note while programming

⊛ Refer to your machine manual!

Possible datum shift values in the rotary axes will be specified by your machine tool builder in the **presetToAlignAxis** parameter (no. 300203).

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

11.3 DATUM SHIFT with datum tables (Cycle 7, ISO: G53)

Effect

Datum tables are used for:

- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within an NC program, you can either program datum points directly in the cycle definition or call them from a datum table.
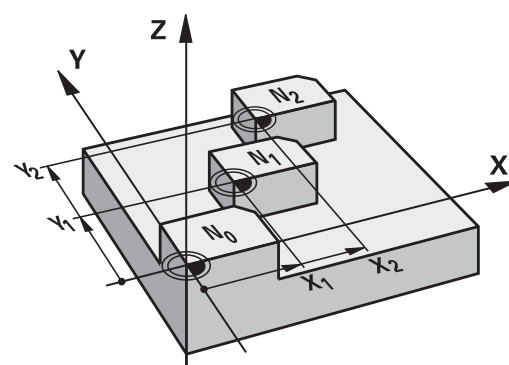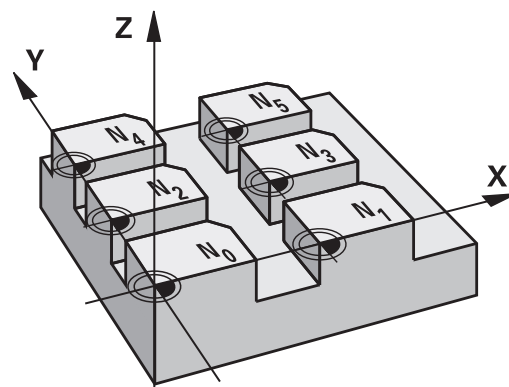
Resetting

- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.
- To shift the datum back to the coordinates X=0, Y=0 etc., directly call a cycle definition

Status displays

In the additional status display, the following data from the datum table is shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

Please note while programming:

> **i**　Datums from a datum table **always and exclusively** reference the current preset.
>
> If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.
>
> In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.
>
> If you work without **SEL TABLE,** then you must activate the desired datum table before the test run or the program run (this applies also to the program run):
>
> - Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table now has the status S
> - Use the file management to select the desired table for the **Program run, single block** and **Program run, full sequence** operating modes: The table now has the status M
>
> The coordinate values from datum tables are only effective with absolute coordinate values.
>
> New lines can only be inserted at the end of the table.
>
> If you create datum tables, the file name has to start with a letter.

Cycle parameters
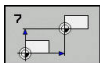
▶ **Displacement**: Enter the number of the datum from the datum table or in a Q parameter. If you enter a Q parameter, the control activates the datum number entered in the Q parameter. Input range: 0 to 9999

Example

77 CYCL DEF 7.0 DATUM SHIFT
78 CYCL DEF 7.1 #5

Selecting a datum table in the part program

With the **SEL TABLE** function, you select the datum table from which the control takes the datums:

PGM
CALL

▶ To select the functions for program call, press the **PGM CALL** key

DATUM
TABLE

▶ Press the **DATUM TABLE** soft key

▶ Enter the complete path name of the datum table or select the file with the **SELECT** soft key. Confirm your input with the **END** key.

> **ⓘ** Program a **SEL TABLE** block before Cycle 7 Datum Shift.
>
> A datum table selected with **SEL TABLE** remains active until you select another datum table with **SEL TABLE** or through **PGM MGT**.

Editing the datum table in the Programming mode of operation

> **ⓘ** After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise, the change will not be taken into account when the NC program is executed.

Select the datum table in the **Programming** mode of operation**Programming**

PGM
MGT

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

▶ Display the datum tables: Press the **SELECT TYPE** and **SHOW .D** soft keys

▶ Select the desired table or enter a new file name.

▶ Edit the file. The functions in the soft-key row include:

Soft key	Function
BEGIN	Select the beginning of the table
END	Select the table end
PAGE	Go to previous page
PAGE	Go to next page
INSERT LINE	Insert line (only possible at the end of table)
DELETE LINE	Delete line
FIND	Find
BEGIN LINE	Move the cursor to the beginning of the line
END LINE	Move the cursor to the end of the line
COPY FIELD	Copy the current value
PASTE FIELD	Insert the copied value
APPEND N LINES AT END	Add the entered number of lines (datums) to the end of the table

Configuring a datum table

If you do not wish to define a datum for an active axis, press the **DEL** key. Then the control clears the numerical value from the corresponding input field.

> ℹ️ You can change the properties of tables. Enter code number 555343 in the MOD menu. The control then displays the **EDIT FORMAT** soft key if a table is selected. When you press this soft key, the control opens a pop-up window where the properties are shown for each column of the selected table. Any changes you make only affect the open table.

Leaving a datum table

Select a different type of file in file management. Select the desired file.

NOTICE

Danger of collision!

The control considers changes in a datum table only when the values are saved.

▶ Make sure to confirm any changes made to the table immediately by pressing the **ENT** key
▶ Carefully test the NC program after making a change to the datum table

---

Status displays

In the additional status display, the control shows the values of the active datum shift.

11.4 PRESETTING (Cycle 247, ISO: G247)

Effect

With the presetting cycle you can activate as the new preset a preset defined in the preset table.

After a presetting cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

Status display

In the status display; the control shows the active preset number behind the preset symbol.
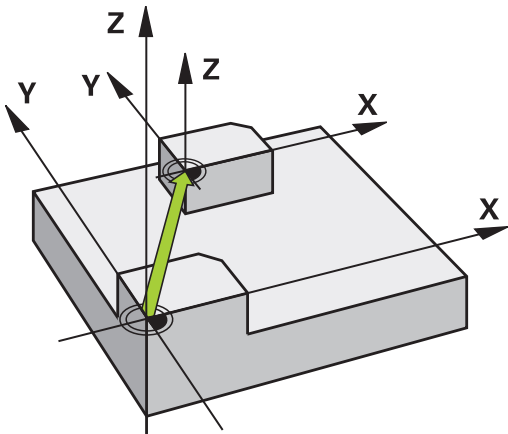
Please note before programming:

> ℹ️ 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** or **Electronic handwheel** operating mode.
>
> Cycle 247 is also effective in the Test Run operating mode.

Cycle parameters

> ▶ **Number for preset?**: Enter the number of the desired preset from the preset table. Alternatively, you can press the **SELECT** soft key and directly select the desired preset from the preset table. Input range: 0 to 65535

Status displays

In the additional status display (**STATUS POS.**) the control shows the active preset number behind the **Preset** dialog.

Example

13 CYCL DEF 247 PRESETTING	
Q339=4	;PRESET NUMBER

11.5 MIRRORING (Cycle 8, ISO: G28)

Effect

The control can machine the mirror image of a contour in the working plane.

Mirroring becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed (except in 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.
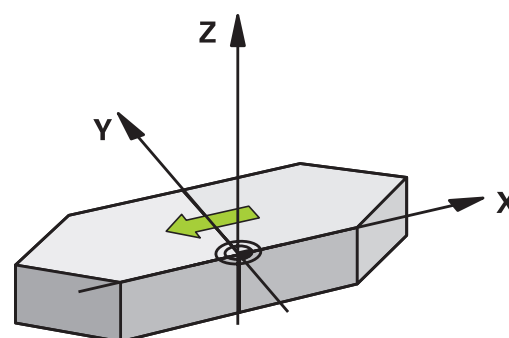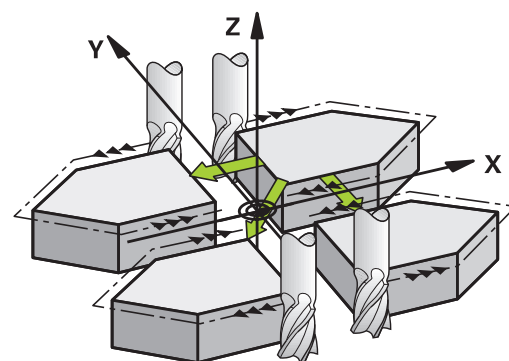
Resetting

Program the MIRROR IMAGE cycle once again with **NO ENT**.

Please note while programming:

> **i** If you work 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
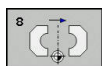
▶ **Mirror image axis?**: Enter the axis to be mirrored. You can mirror all axes—including rotary axes—except for the spindle axis and its associated secondary axis. You can enter up to three axes. Input range: up to three NC axes **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example

79 CYCL DEF 8.0 MIRRORING
80 CYCL DEF 8.1 X Y Z

11.6 ROTATION (Cycle 10, ISO: G73)

Effect

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 effective in the Positioning with Manual Data Input operating mode. 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
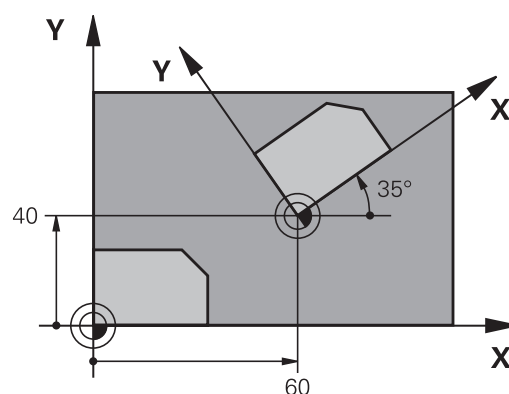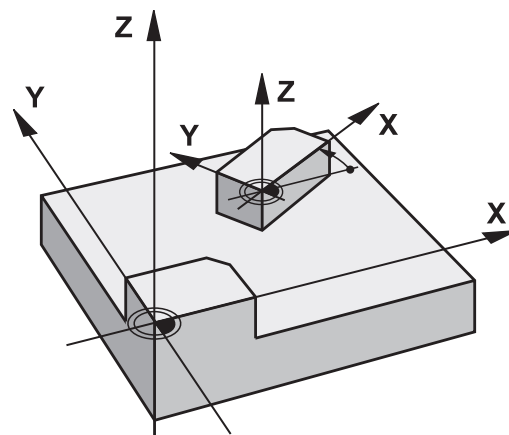
Resetting

Program the ROTATION cycle once again with a rotation angle of 0°.

Please note while programming:

	Cycle 10 cancels an active radius compensation. If necessary, reprogram the radius compensation.
i	After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

Cycle parameters
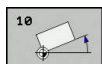
▶ **Rotation**: Enter the angle of rotation in degrees (°). Input range: -360.000° to +360.000° (absolute or incremental)

Example

12 CALL LBL 1
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 10.0 ROTATION
17 CYCL DEF 10.1 ROT+35
18 CALL LBL 1

11.7   SCALING (Cycle 11, ISO: G72)

Effect

The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The factor defined for SCALING becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- all three coordinate axes at the same time
- dimensions in cycles

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

Reduction: SCL less than 1 (down to 0.000 001)

Resetting

Program the SCALING cycle once again with a scaling factor of 1.
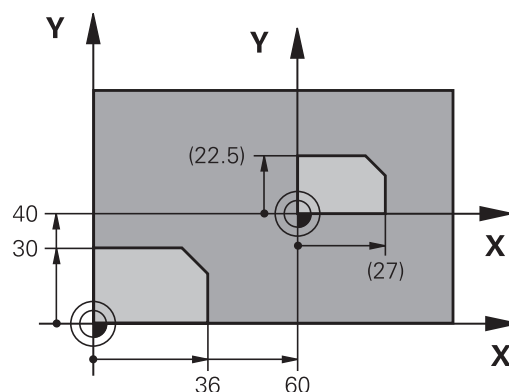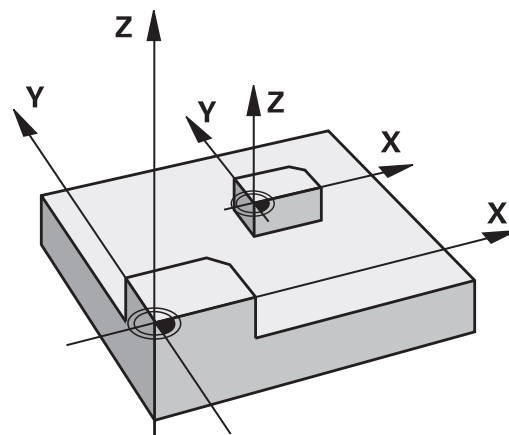
Cycle parameters
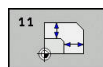
▶ **Factor?**: Enter the scaling factor SCL. The control multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range: 0.000001 to 99.999999

Example

11 CALL LBL 1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1

11.8 AXIS-SPECIFIC SCALING (Cycle 26)

Effect

With Cycle 26, you can account for shrinkage and oversize factors for each axis.

The factor defined for SCALING becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active scaling factor is shown in the additional status display.

Resetting

Program the SCALING cycle once again with a scaling factor of 1 for the corresponding axis.
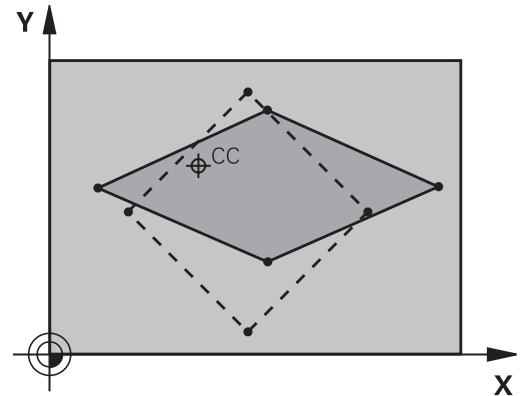
Please note while programming:

🛈 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.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.

Cycle parameters
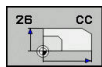
▶ **Axis and factor**: Select the coordinate axis/axes via soft key. Enter the factor(s) for axis-specific enlargement or reduction. Input range: 0.000001 to 99.999999

▶ **Center coordinates**: Enter the center of the axis-specific enlargement or reduction. Input range -99999.9999 to 99999.9999
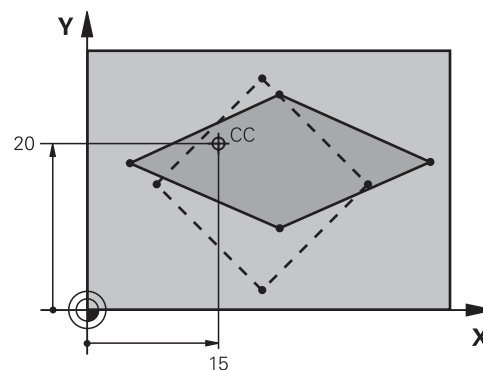
Example

25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL 1

11.9 WORKING PLANE (Cycle 19, ISO: G80, software option 1)

Effect

In Cycle 19 you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to three rotations (spatial angles) of the **machine-based** coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.

> ℹ️ Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.

If you program the position of the working plane via spatial angles, the control will calculate the required angle positions of the tilted axes automatically and will store these in the Q120 (A axis) to Q122 (C axis) parameters. If two solutions are possible, the control will choose the shorter path from the current position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The control first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it has been defined in the NC program. As soon as you move an axis in the tilted system, the compensation for this specific axis will be activated. You must move all axes to activate compensation for all axes.

If you set the **Tilting program run** parameters to **Active** in the Manual Operation mode, the angular value entered in this menu is overwritten by Cycle 19 WORKING PLANE.
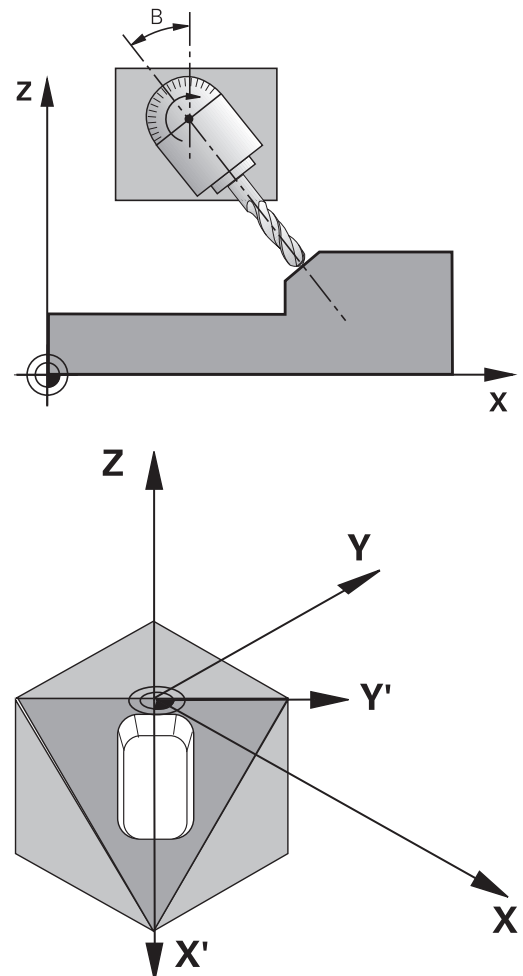
Please note while programming:

⚙   The **Tilt working plane** functions are interfaced to the control and the machine tool by the machine tool builder.

     The machine tool builder also specifies whether the programmed angles are interpreted as coordinates of the rotary axes (axis angles) or as angular components of a tilted plane (spatial angles).

ⓘ   Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

     The working plane is always tilted around the active datum.

     If you use the Cycle 19 while M120 is active, the control automatically cancels the radius compensation, which also cancels the M120 function.

     In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

Cycle parameters
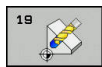
▶ **Rotary axis and angle?**: Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys. Input range: -360.000 to 360.000

If the control automatically positions the rotary axes, you can enter the following parameters:

▶ **Feed rate? F=**: Traversing speed of the rotary axis during automatic positioning. Input range 0 to 99999.999

▶ **Set-up clearance?** (incremental): The control positions the tilting head in such a way that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. Input range: 0 to 99999.9999
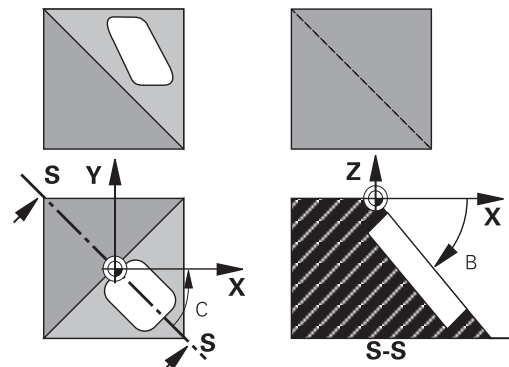
Reset

To reset the tilt angles, redefine the WORKING PLANE cycle. Enter an angular value of 0° for all rotary axes. Then, redefine the WORKING PLANE cycle. Confirm the dialog prompt by pressing the **NO ENT** key. This disables the function.

Positioning the axes of rotation

⚙ Refer to your machine manual.

The machine tool builder determines whether Cycle 19 positions the axes of rotation automatically or whether they must be positioned manually in the NC program.

Manual positioning of rotary axes

If Cycle 19 does not position the rotary axes automatically, you need to position them in a separate L block following the cycle definition.

If you use axis angles, you can define the axis values right in the L block. If you use spatial angles, then program the Q parameters **Q120** (A axis value), **Q121** (B axis value) and **Q122** (C axis value) according to Cycle 19.

ℹ For manual positioning, always use the rotary axis positions stored in Q parameters Q120 to Q122.

Avoid using functions, such as M94 (modulo rotary axes), in order to avoid discrepancies between the actual and nominal positions of rotary axes in multiple definitions.

Example

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the spatial angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0	
14 L A+Q120 C+Q122 R0 F1000	Position the rotary axes by using values calculated by Cycle 19
15 L Z+80 R0 FMAX	Activate compensation for the spindle axis
16 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle 19:

- The control can position only closed-loop axes.
- To position the tilted axes, you must enter a feed rate and a set-up clearance, in addition to the tilting angles, when defining the cycle.
- Use only preset tools (the full tool length must be defined).
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting
- The control performs tilting at the last programmed feed rate. The maximum feed rate depends on the complexity of the swivel head (tilting table).

Example

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0 F5000 ABST50	Also define the feed rate and the clearance
14 L Z+80 R0 FMAX	Activate compensation for the spindle axis
15 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Position display in a tilted system

On activation of Cycle 19, the displayed positions (**ACTL** and **NOML**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle 19.

Monitoring of the working space

The control monitors only those axes in the tilted coordinate system that are moved. Where applicable, the control displays an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system.

It is also possible to position the axes using straight-line blocks that reference the machine coordinate system (NC blocks with M91 or M92), if the working plane is tilted. Restrictions:

- Positioning is without length compensation.
- Positioning is done without length compensation.
- Tool radius compensation is not allowed.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is tilted about the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after having activated Cycle 19, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

    1st Activate datum shift

    2nd Activate tilting function

    3rd Activate rotation

    ...

    Workpiece machining

    ...

    1st Reset rotation

    2nd Reset tilting function

    3rd Reset datum shift

Procedure for working with Cycle 19 WORKING PLANE

1 Create the NC program

▶ Define the tool (not required if TOOL.T is active), and enter the full tool length.

▶ Call the tool.

▶ Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece or clamping devices during tilting.

▶ If required, position the tilt axis or axes with an L block to the appropriate angular value(s) (depending on a machine parameter).

▶ Activate datum shift if required.

▶ Define Cycle 19 WORKING PLANE; enter the angular values for the tilt axes.

▶ Traverse all principal axes (X, Y, Z) to activate compensation.

▶ Write the program as if the machining process were to be executed in a non-tilted plane.

▶ If required, define Cycle 19 WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle 19—you can define the new angular values directly.

▶ Reset Cycle 19 WORKING PLANE by programming 0° for all tilt axes.

▶ Disable the WORKING PLANE function; redefine Cycle 19. Confirm the dialog prompt with **NO ENT**.

▶ Reset datum shift if required.

▶ Position the tilt axes to the 0° position if required.

2 Clamp the workpiece

3 Presetting

■ Manually by touch-off

■ Controlled by a HEIDENHAIN 3-D touch probe

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

■ Automatically with a HEIDENHAIN 3-D touch probe
Further information: "Touch Probe Cycles: Automatic Presetting", Page 405)

4 Start the NC program in the Program Run, Full Sequence operating mode

5 Manual Operation mode

Use the 3-D ROT soft key to set the TILT WORKING PLANE function to INACTIVE. Enter an angular value of 0° for each rotary axis in the menu.

11.10 Programming Examples

Example: coordinate transformation cycles

Program run

- Program the coordinate transformations in the main program
- Machining within a subprogram
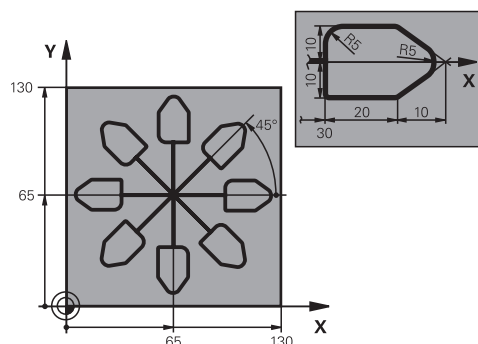
0 BEGIN PGM COTRANS MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+130 X+130 Z+0	
3 TOOL CALL 1 Z S4500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center
6 CYCL DEF 7.1 X+65	
7 CYCL DEF 7.2 Y+65	
8 CALL LBL 1	Call milling operation
9 LBL 10	Set label for program section repeat
10 CYCL DEF 10.0 ROTATION	Rotate by 45° (incremental)
11 CYCL DEF 10.1 IROT+45	
12 CALL LBL 1	Call milling operation
13 CALL LBL 10 REP 6/6	Return jump to LBL 10; repeat the milling operation six times
14 CYCL DEF 10.0 ROTATION	Reset the rotation
15 CYCL DEF 10.1 ROT+0	
16 CYCL DEF 7.0 DATUM SHIFT	Reset datum shift
17 CYCL DEF 7.1 X+0	
18 CYCL DEF 7.2 Y+0	
19 L Z+250 R0 FMAX M2	Retract the tool, end program
20 LBL 1	Subprogram 1
21 L X+0 Y+0 R0 FMAX	Define milling operation
22 L Z+2 R0 FMAX M3	
23 L Z-5 R0 F200	
24 L X+30 RL	
25 L IY+10	
26 RND R5	
27 L IX+20	
28 L IX+10 IY-10	

29 RND R5	
30 L IX-10 IY-10	
31 L IX-20	
32 L IY+10	
33 L X+0 Y+0 R0 F5000	
34 L Z+20 R0 FMAX	
35 LBL 0	
36 END PGM COTRANS MM	

12

Cycles: Special Functions

12.1 Fundamentals

Overview

The control provides the following cycles for the following special purposes:

Soft key	Cycle	Page
	9 DWELL TIME	325
	12 Program call	326
	13 Oriented spindle stop	327
	32 TOLERANCE	328
	225 ENGRAVING of texts	332
	232 FACE MILLING	338
	239 ASCERTAIN THE LOAD	343

12.2 DWELL TIME (Cycle 9, ISO: G04)

Function

Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.

The cycle becomes effective as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Example

89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5

Cycle parameters

▶ **Dwell time in seconds**: Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

12.3   PROGRAM CALL (Cycle 12, ISO: G39)

Cycle function

NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.
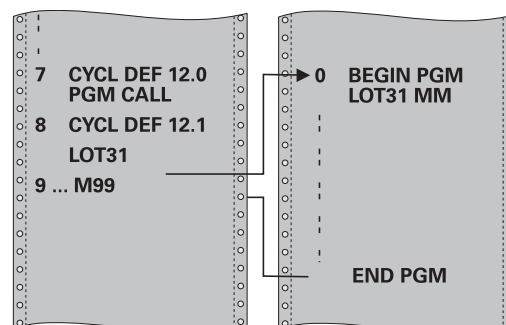
Please note while programming:

> **i**   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.
>
> As a rule, Q parameters are globally effective when called with Cycle 12. So please note that changes to Q parameters in the called NC program can also influence the calling NC program.

Cycle parameters

> ▶ **Program name**: Enter the name of the NC program and, if necessary, the path where it is located, or
> ▶ Activate the file select dialog with the **SELECT** soft key. Select the NC program to be called.

Call the NC program with:
- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare program 50.h as a cycle and call it with M99

55 CYCL DEF 12.0 PGM CALL
56 CYCL DE 12.1 PGM TNC: \KLAR35\FK1\50.H
57 L X+20 Y+50 FMAX M99

12.4 SPINDLE ORIENTATION (Cycle 13, ISO: G36)

Cycle function

🔩 Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The control can control the main machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

With M19 or M20, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program M19 or M20 without having defined Cycle 13 beforehand, the control positions the main spindle at an angle that has been set by the machine tool builder.

More information: machine tool manual.
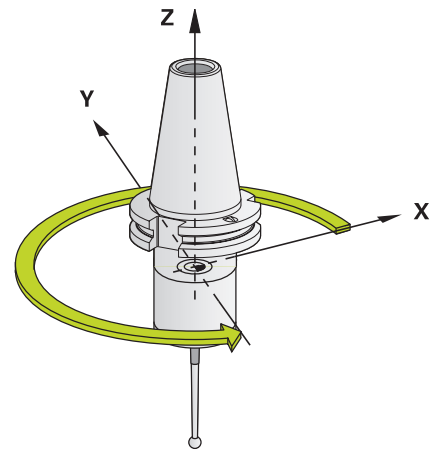
Example

93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180

Please note while programming:

ℹ️ 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

▶ **Angle of orientation**: Enter the angle with reference to the angle reference axis of the working plane. Input range: 0.0000° to 360.0000°

12.5 TOLERANCE (Cycle 32, ISO: G62)

Cycle function

⚙️ Machine and control must be specially prepared by the machine tool builder 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, inasmuch 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 tool builder has implemented these features.

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 post processor (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 if in Cycle 32 you choose a tolerance value between 110-% and 200-% of the CAM chord error.
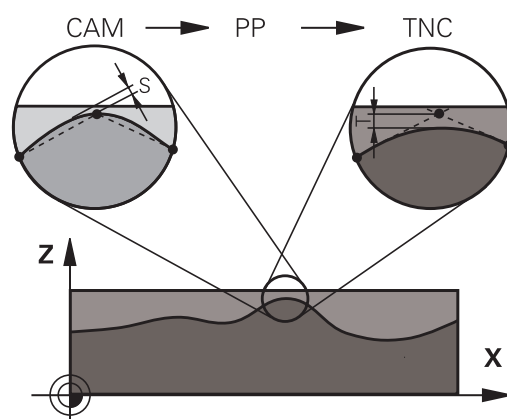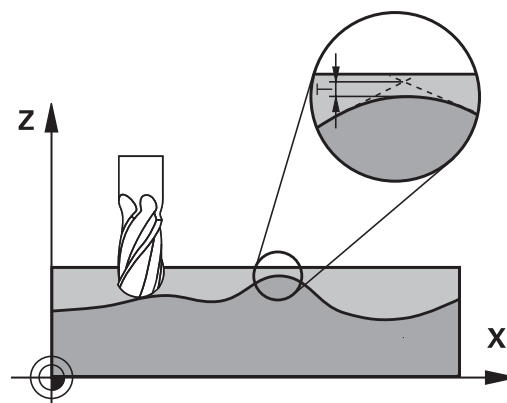
Please note while programming!

ℹ 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.

Cycle 32 is DEF-active which means that it becomes effective as soon as it is defined in the NC program.

The control resets Cycle 32 if you do one of the following:

- Redefine Cycle 32 it and confirm the dialog prompt for the **tolerance value** with **NO ENT**.
- Select a new NC program with the **PGM MGT** key.

After you have reset Cycle 32, the control reactivates the tolerance that was predefined by the machine parameters.

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 tool builder).

If Cycle 32 is active, the control shows the parameters defined for Cycle 32 on the **CYC** tab of the additional status display.

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 consistent. Additionally, in you can set a higher rotational axis tolerance **TA** (e.g. between 1° and 3°) for an even more constant feed-rate curve at the tool reference 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

R: Major radius in mm

T_{32}: Machining tolerance in mm

Cycle parameters
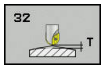
▶ **Tolerance value T**: Permissible contour deviation in mm (or inches with inch programming). Input range: 0.0000 to 10.0000
>0: If you enter a value greater than zero, the control will use the maximum permissible deviation you specify
0: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine tool builder.

▶ **HSC MODE, Finishing=0, Roughing=1**: Activate filter:

■ Input value 0: **Milling with increased contour accuracy**. The control uses internally defined finishing filter settings.

■ Input value 1: **Milling at an increased feed rate**. The control uses internally defined roughing filter settings.

▶ **Tolerance for rotary axes TA**: Permissible position error of rotary axes in degrees when M128 is active (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 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. Input range: 0.0000 to 10.0000
>0: If you enter a value greater than zero, the control will use the maximum permissible deviation you specify.
0: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine tool builder.

Example

95 CYCL DEF 32.0 TOLERANCE
96 CYCL DEF 32.1 T0.05
97 CYCL DEF 32.2 HSC-MODE:1 TA5

12.6 ENGRAVING (Cycle 225, ISO: G225)

Cycle run

This cycle is used to engrave texts on a flat surface of the workpiece. The texts can be arranged in a straight line or along an arc.

1 The control positions the tool in the working plane to the starting point of the first character

2 The tool plunges perpendicularly to the engraving floor and mills the character. The control retracts the tool to the set-up clearance between the characters when required. After machining the character, the tool is at the set-up clearance above the workpiece surface.

3 This process is repeated for all characters to be engraved.

4 Finally, the control retracts the tool to the 2nd set-up clearance.
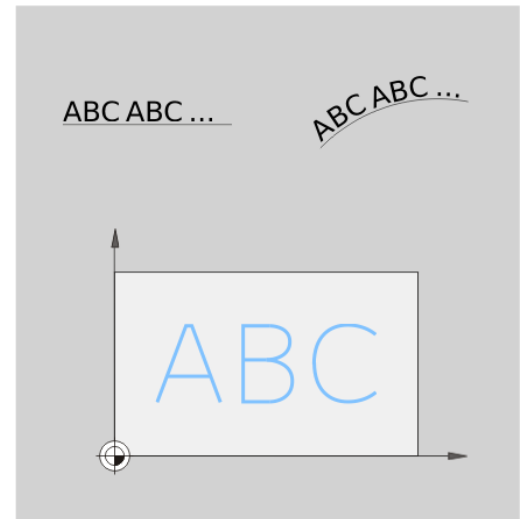
Please note while programming:

> **i** 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.
>
> When engraving on a circular arc, the starting point is at bottom left, above the first character to be engraved. (With older software versions, the tool may be pre-positioned at the center of the circle.)

Cycle parameters

▶ **QS500 Engraving text?**: Text to be engraved, in quotation marks. Maximum input: 255 characters. Assignment of a string variable through the **Q** key of the numerical keypad. The **Q** key on the alphabetic keyboard represents normal text input. see "Engraving system variables", Page 336

▶ **Q513 Character height?** (absolute): Height of the characters to be engraved in mm. Input range: 0 to 99999.9999

▶ **Q514 Character spacing factor?**: The font used is a proportional font. Each character has its own width, which is engraved correspondingly by the control if you program Q514=0. If Q514 is not equal to 0, the control scales the space between the characters. Input range: 0 to 9.9999

▶ **Q515 Font?**: By default, the control uses the **DeJaVuSans** font.

▶ **Q516 Text on a line/on an arc(0/1)?**:
Engrave the text in a straight line: Input = 0
Engrave the text on an arc: Input = 1
Engrave the text on an arc, circumferentially (not necessarily legible from below): Input = 2

▶ **Q374 Angle of rotation?**: Center angle if the text is to be engraved on an arc. Engraving angle when text is in a straight line. Input range: -360.0000 to +360.0000°

▶ **Q517 Radius of text on an arc?** (absolute): Radius of the arc in mm on which the control is to engrave the text Input range: 0 to 99999.9999

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q201 Depth?** (incremental): Distance between workpiece surface and engraving floor

▶ **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**

▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**

$$a = x * Q514$$
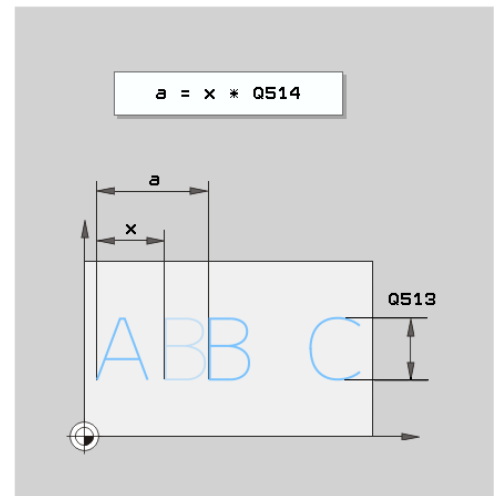
Example

62 CYCL DEF 225 ENGRAVING	
QS500="A"	;ENGRAVING TEXT
Q513=10	;CHARACTER HEIGHT
Q514=0	;SPACE FACTOR
Q515=0	;FONT
Q516=0	;TEXT ARRANGEMENT
Q374=0	;ANGLE OF ROTATION
Q517=0	;CIRCLE RADIUS
Q207=750	;FEED RATE FOR MILLNG
Q201=-0.5	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q200=2	;SET-UP CLEARANCE
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q367=+0	;TEXT POSITION
Q574=+0	;TEXT LENGTH

▶ **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

▶ **Q367 Reference for text position (0-6)?** Enter here the reference for the position of the text. Depending on whether the text will be engraved on an arc or a straight line (parameter Q516), the following values can be entered:
If engraved on an arc, the text position refers to the following point:
0 = Center of the circle
1 = Bottom left
2 = Bottom center
3 = Bottom right
4 = Top right
5 = Top center
6 = Top left
If engraved on a straight line, the text position refers to the following point:
0 = Bottom left
1 = Bottom left
2 = Bottom center
3 = Bottom right
4 = Top right
5 = Top center
6 = Top left

▶ **Q574 Maximum text length?** (mm/inch): Enter the maximum text length here. 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 is greater than zero, 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.

Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers:

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

> ℹ️ The control uses the special characters **%** and ****
> for special functions. f 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:

Algebraic sign	Input
ä	%ae
ö	%oe
ü	%ue
Ä	%AE
Ö	%OE
Ü	%UE
ß	%ss
ø	%D
@	%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:

Character	Input
Line break	\n
Horizontal tab (the tab width is permanently set to 8 characters)	\t
Vertical tab (the tab width is permanently set to one line)	\v

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 or time. Do do so, enter **%time<x>**. **<x>** defines the format, e.g. 08 for DD.MM.YYYY. (Identical to the **SYSSTR ID321** function)

i	Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. **time08**.

Character	Input
DD.MM.YYYY hh:mm:ss	%time00
D.MM.YYYY h:mm:ss	%time01
D.MM.YYYY h:mm	%time02
D.MM.YY h:mm	%time03
YYYY-MM-DD hh:mm:ss	%time04
YYYY-MM-DD hh:mm	%time05
YYYY-MM-DD h:mm	%time06
YY-MM-DD h:mm	%time07
DD.MM.YYYY	%time08
D.MM.YYYY	%time09
D.MM.YY	%time10
YYYY-MM-DD	%time11
YY-MM-DD	%time12
hh:mm:ss	%time13
h:mm:ss	%time14
h:mm	%time15

Engraving the counter reading

You can engrave the current counter reading found in the MOD menu with Cycle 225.

For this purpose program Cycle 225 as usual and enter e.g. the following for the text to be engraved: **%count2**

The number after **%count** indicates how many digits the control will engrave. The maximum is of nine digits.

Example: If you program **%count9** in the cycle with a momentary counter reading of 3, the control will engrave the following: 000000003

> **ⓘ**　In the Test Run operating mode, the control only simulates the counter reading you specified directly in the NC program. The counter reading from the MOD menu is not taken into account.
>
> In the SINGLE BLOCK and FULL SEQ. and Program Run, Single Block operating modes, the control will take the counter reading from the MOD menu into account.

12.7 FACE MILLING (Cycle 232, ISO: G232, software option 19)

Cycle run

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

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 machining plane and then in the spindle axis. Otherwise, it first moves it to the 2nd set-up clearance and then in the machining plane. The starting point in the machining 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.
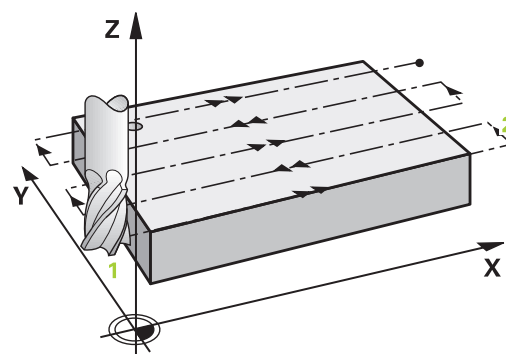
Strategy Q389=1

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.
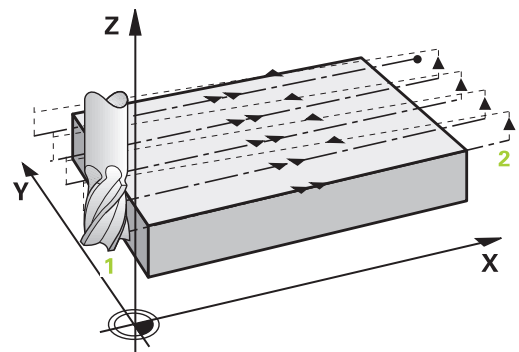
Please note while programming:

> ⓘ Enter **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.
>
> If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
>
> Program Q227 greater than Q386. The control will otherwise display an error message.

Cycle parameters

▶ **Q389 Machining strategy (0/1/2)?**: Determine how the control should machine the surface:
0: Meander machining, stepover at the positioning feed rate outside the surface being machined
1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
2: Line-by-line machining, retraction and stepover at the positioning feed rate

▶ **Q225 Starting point in 1st axis?** (absolute): Starting point coordinate of the surface to be machined in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q226 Starting point in 2nd axis?** (absolute): Starting point coordinate of the surface to be machined in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range: -99999.9999 to 99999.9999

▶ **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis to which the surface is to be face-milled. Input range: -99999.9999 to 99999.9999

▶ **Q218 First side length?** (incremental): Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path referencing the **starting point in the 1st axis**. Input range: -99999.9999 to 99999.9999

▶ **Q219 Second side length?** (incremental): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **STARTNG PNT 2ND AXIS**. Input range: -99999.9999 to 99999.9999

▶ **Q202 Maximum plunging depth?** (incremental): **Maximum** infeed per cut. The control calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. Input range: 0 to 99999.9999

▶ **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range: 0 to 99999.9999
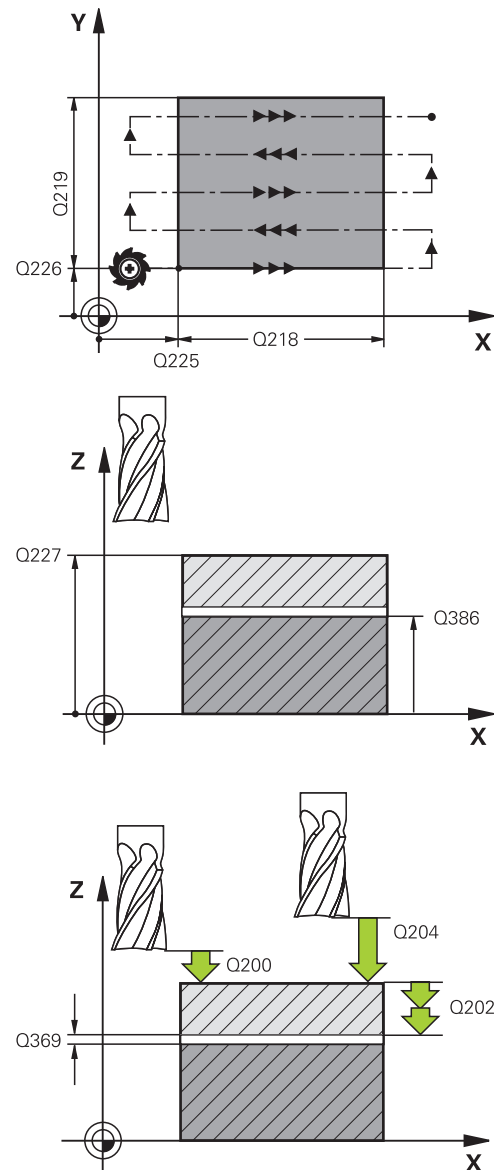
▶ **Q370 Max. path overlap factor?**: **Maximum** stepover factor k. The control calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. cutter radius when using a face-milling cutter), the control reduces the stepover accordingly. Input range: 0.1 to 1.9999

▶ **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**

▶ **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**

▶ **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (Q389=1), the control uses the cross feed rate for milling Q207. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**

▶ **Q200 Set-up clearance?** (incremental): 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 the set-up clearance above the current plunging depth to the starting point of the next pass. Input range: 0 to 99999.9999

▶ **Q357 Safety clearance to the side?** (incremental) Parameter Q357 affects the following situations: **Approaching the first plunging depth:** Q357 is the lateral distance between tool and workpiece **Roughing with the Q389=0-3 roughing strategies:** The surface to be machined is extended in **Q350** MILLING DIRECTION by the value from Q357 if no limit has been set in that direction **Side finishing:** The paths will be extended by Q357 in **Q350** MILLING DIRECTION. Input range: 0 to 99999.9999

▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

Example

71 CYCL DEF 232 FACE MILLING	
Q389=2	;STRATEGY
Q225=+10	;STARTNG PNT 1ST AXIS
Q226=+12	;STARTNG PNT 2ND AXIS
Q227=+2.5	;STARTNG PNT 3RD AXIS
Q386=-3	;END POINT 3RD AXIS
Q218=150	;FIRST SIDE LENGTH
Q219=75	;2ND SIDE LENGTH
Q202=2	;MAX. PLUNGING DEPTH
Q369=0.5	;ALLOWANCE FOR FLOOR
Q370=1	;MAX. OVERLAP
Q207=500	;FEED RATE FOR MILLNG
Q385=800	;FINISHING FEED RATE
Q253=2000	;F PRE-POSITIONING
Q200=2	;SET-UP CLEARANCE
Q357=2	;CLEARANCE TO SIDE
Q204=2	;2ND SET-UP CLEARANCE

12.8 ASCERTAIN THE LOAD (Cycle 239, ISO: G239, software option 143)

Cycle run

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 return a certain path. 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 tool builder.

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.

Parameter Q570 = 0

1 There is no physical movement of the axes.
2 The control resets the LAC.
3 The TNC activates feedforward parameters and, if applicable, controller parameters that ensure a safe movement of the axes concerned, regardless of the 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 tool builder.
3 The feedforward and controller parameters determined by the control **depend** on the current load.
4 The control activates the parameters determined.

Please note while programming:

⚙️ The machine must have been prepared by the machine tool builder for this cycle.

Cycle 239 works only together with Option 143 LAC (Load Adaptive Control).

NOTICE

Danger of collision!

The cycle can execute extensive movements in several axes at rapid traverse!

- ▶ Ask your machine manufacturer about the type and extent of Cycle 239 before using it
- ▶ Before the cycle starts, the control moves to a safe position, if applicable. The machine tool builder 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.

ℹ️ Cycle 239 becomes effective immediately after its definition.

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 procedure will be performed.

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

▶ **Q570 Load (0 = Delete/1 = Ascertain)?**: Specify whether the control should perform an LAC (Load adaptive control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters should 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.
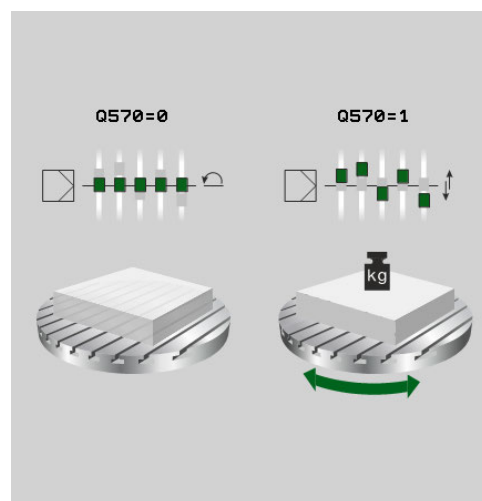
Example

62 CYCL DEF 239 ASCERTAIN THE LOAD
Q570=+0   ;LOAD ASCERTATION

12.9 THREAD CUTTING (Cycle 18, ISO: G18 , software option 19)

Cycle run

Cycle **18** THREAD CUTTING moves the tool with servo-controlled spindle from the momentary position with active speed to the specified depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.

Please note while programming:

> ℹ️   Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:
>
> - **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (speed override is not active). The Control then adapts the spindle speed as required.
> - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
> - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
> - **limitSpindleSpeed** (no. 113604): Spindle speed limit
>   True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time
>   False: (Limiting not active)
>
> The spindle speed potentiometer is inactive.
>
> Before calling this cycle, be sure to program a spindle stop! (For example with M5.) The control automatically activates spindle rotation at the start of the cycle and deactivates it at the end.
>
> The algebraic sign for the cycle parameter "thread depth" determines the working direction.

NOTICE

Danger of collision!

A collision may occur if you do not program pre-positioning before calling Cycle 18. Cycle 18 does not perform approach and departure motion.

- ▶ Pre-position the tool before the start of the cycle.
- ▶ The tool moves from the current position to the entered depth after the cycle is called

---

NOTICE

Danger of collision!

If the spindle was switched on before calling the cycle, Cycle 18 switches the spindle off and the cycle works with a stationary spindle! Cycle 18 switches the spindle on again at the end if it was switched on before cycle start.

- ▶ Before starting this cycle, be sure to program a spindle stop! (For example with M5.)
- ▶ At the end of Cycle 18, the control restores the spindle to its state at cycle start. If the spindle was switched off before this cycle, the control will switch it off again at the end of Cycle 18.

Cycle parameters
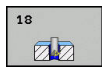
▶ Boring depth (incremental): Enter the thread depth based on the current position. Input range: -99999 ... +99999

▶ Thread pitch: Enter the pitch of the thread. The algebraic sign entered here differentiates between right-hand and left-hand threads:
+ = right-hand thread (M3 with negative hole depth)
- = left-hand thread (M4 with negative hole depth)
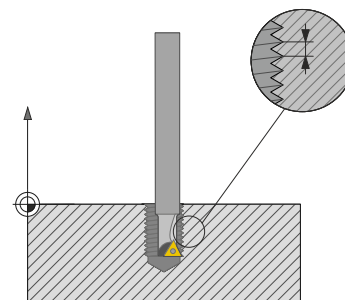
Example

25 CYCL DEF 18.0 THREAD CUTTING
26 CYCL DEF 18.1 DEPTH = -20
27 CYCL DEF 18.2 PITCH = +1

13

Using Touch Probe Cycles

13.1 General information about touch probe cycles

ℹ️	HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

⚙️	The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

Method of function

Whenever the control runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Further information: "Before You Start Working with Touch Probe Cycles", Page 353

When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the control displays an error message (distance: **DIST** from touch probe table).

Consideration of a basic rotation in the Manual Operation mode

During probing, the control considers an active basic rotation and approaches the workpiece at an angle.

Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes

In the **Manual operation** and **Electronic handwheel** modes, the control provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensate workpiece misalignment
- Set presets
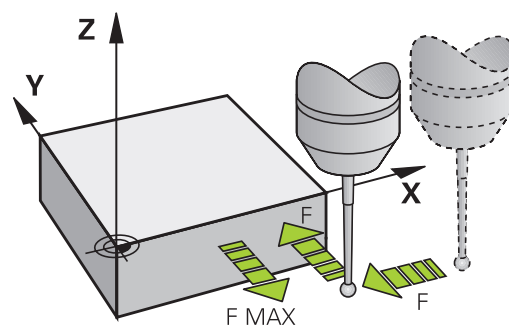
Touch probe cycles for automatic operation

Besides the touch probe cycles, which you can use in the Manual and El. Handwheel modes, the control provides numerous cycles for a wide variety of applications in automatic mode:

- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Presetting
- Automatic workpiece inspection
- Automatic tool measurement

You can program the touch probe cycles in the **Programming** operating mode via the **TOUCH PROBE** key. Like the most recent fixed cycles, touch probe cycles with numbers greater than 400 use Q parameters as transfer parameters. Parameters with the same function that the control requires in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the control shows a graphic during cycle definition. In the graphic, the parameter that needs to be entered is highlighted (see figure right).
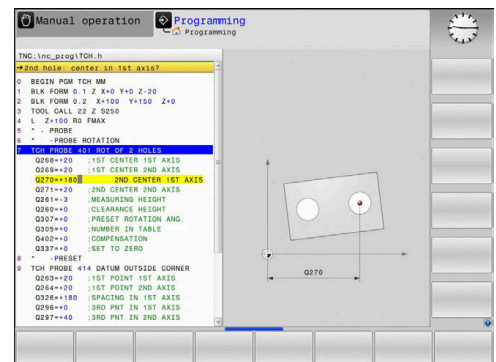
Defining the touch probe cycle in Programming mode of operation

TOUCH PROBE

▶ The soft-key row shows all available touch probe functions divided into groups.

PRESET

▶ Select the desired cycle group, e.g. presetting. Cycles for automatic tool measurement are available only if your machine has been prepared for them.

410

▶ Select a cycle, point e.g. presetting at pocket center. The control initiates the programming dialog and asks for all required input values. At the same time, a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.

▶ Enter all parameters requested by the control and confirm each entry with the ENT key.

▶ The control closes the dialog when all required data has been entered.

Soft key	Measuring cycle group	Page
ROTATION	Cycles for automatic measurement and compensation of workpiece misalignment	359
PRESET	Cycles for automatic workpiece presetting	406
MEASURING	Cycles for automatic workpiece inspection	464
SPECIAL CYCLES	Special cycles	508
CALIBRATE TS	Calibrate TS	508
KINEMATICS	Kinematics	531
TT CYCLES	Cycles for automatic tool measurement (enabled by the machine manufacturer)	564

NC blocks

5 TCH PROBE 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=+0	;PRESET

13.2 Before You Start Working with Touch Probe Cycles

To make it possible to cover the widest possible range of applications, machine parameters enable you to determine the behavior common to all touch probe cycles.

Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the range defined in **DISST**, the control will issue an error message.
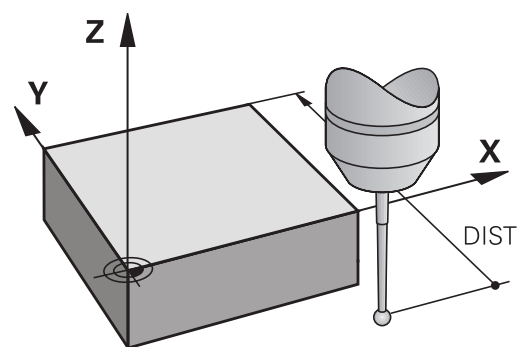
Set-up clearance to touch point: SET_UP in touch probe table

In **SET_UP** you define how far from the defined (or calculated) touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to **SET_UP**.
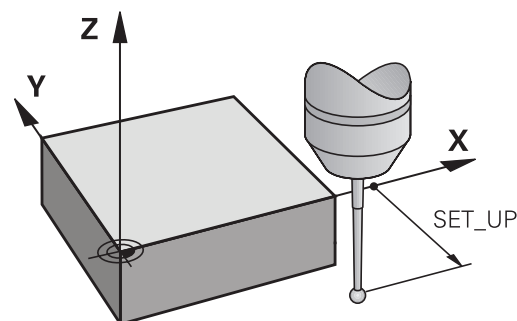
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table

To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

> **i** If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch trigger probe, probing feed rate: F in touch probe table

In **F**, you define the feed rate at which the control is to probe the workpiece.

F can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).

The feed rate potentiometer may be effective with touch probe cycles. The machine tool builder defines the required settings. (the parameter **overrideForMeasure** (No. 122604) must be appropriately configured.)

Touch trigger probe, rapid traverse for positioning: FMAX

In **FMAX**, you define the feed rate at which the control pre-positions the touch probe and positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table

In **F_PREPOS**, you define whether the control is to position the touch probe at the feed rate defined in FMAX or at rapid traverse.

- Input value = **FMAX_PROBE**: Position at feed rate from **FMAX**
- Input value = **FMAX_MACHINE**: Pre-position at rapid traverse

Executing touch probe cycles

All touch probe cycles are DEF-active. This means that the control runs the cycle automatically as soon as it executes the cycle definition in the program run.

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 1400 to 1499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **8 MIRROR IMAGE**, Cycles **11 SCALING** and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand.

ⓘ Touch probe cycles 408 to 419 and 1400 to 1499 may be run even if a basic rotation is active. Make sure, however, that the basic rotation angle does not change when you use Cycle 7 DATUM SHIFT after the measuring cycle.

The control will additionally check during probing whether the positions of the rotary axes match the tilting angles (3-D ROT), depending on the setting of the optional **chkTiltingAxes** machine parameter (no. 204600). If that is not the case, the control displays an error message.

Touch probe cycles numbered 400 to 499 or 1400 to 1499 position the touch probe according to the following positioning logic:

■ If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to clearance height and then positions it in the working plane to the first touch point.

■ If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the control first positions the touch probe to the first probe point in the working plane, and then in the touch-probe axis directly to the measuring height.

13.3 Touch-probe table

General information

Various data is stored in the touch probe table that defines the probe behavior during the probing process. If you use several touch probes on your machine tool, you can save separate data for each touch probe.

> **i** You can also view and edit the data of the touch probe table in the expanded tool management (option 93).

Editing touch probe tables

To edit the touch probe table, proceed as follows:

▶ Operating mode: Press the **Manual operation** key

▶ Select the touch probe functions: Press the **TOUCH PROBE** soft key. The control displays additional soft keys.

▶ Select the touch probe table: Press the **TCH PROBE TABLE** soft key.
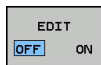
▶ Set the **EDIT** soft key to **ON**.

▶ Using the arrow keys, select the desired setting.

▶ Perform desired changes.

▶ Exit the touch probe table: Press the **END** soft key.
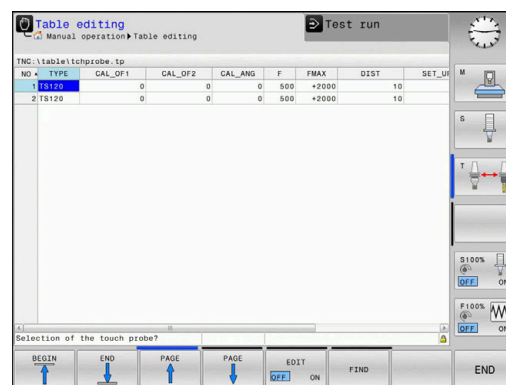
Touch probe data

Abbr.	Inputs	Dialog
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool number	–
TYPE	Selection of the touch probe used	**Selection of the touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis in the principal axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis in the minor axis	**TS center misalignmt. aux. axis? [mm]**
CAL_ANG	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the control will probe the workpiece **F** can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe is pre-positioning and is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within this defined value, the control will issue an error message.	**Maximum measuring range? [mm]**
SET_UP	In **set_up** you define how far from the defined or calculated touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to the **SET_UP** machine parameter.	**Set-up clearance? [mm]**
F_PREPOS	Defining speed with pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-position at rapid? ENT/ NOENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probing process. In this way, the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Probe oriented? Yes=ENT/ No=NOENT**
SERIAL	You need not make an entry in this column. The TNC automatically enters the serial number of the touch probe if the touch probe has an EnDat interface.	**Serial number?**
REACTION	Behavior in case of collision with the touch probe ■ **NCSTOP**: The NC program will be aborted. ■ **EMERGSTOP**: Emergency stop, safe braking of the axes.	**Reaction?**

14

Touch Probe
Cycles: Automatic
Measurement
of Workpiece
Misalignment

14.1 Overview

⚙️ The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Cycle	Page
1420	**1420 PROBING IN PLANE** Automatic measurement using three points. Compensation via basic rotation.	366
1410	**1410 PROBING ON EDGE** Automatic measurement using two points. Compensation via basic rotation or a rotation of the rotary table.	370
1411	**1411 PROBING TWO CIRCLES** Automatic measurement using two cylindrical holes or studs. Compensation via basic rotation or a rotation of the rotary table.	374
400	**400 BASIC ROTATION** Automatic measurement using two points. Compensation via basic rotation.	381
401	**401 ROT OF 2 HOLES** Automatic measurement using two holes. Compensation via basic rotation.	384
402	**402 ROT OF 2 STUDS** Automatic measurement using two studs. Compensation via basic rotation.	388
403	**403 ROT IN ROTARY AXIS** Automatic measurement using two points. Compensation by turning the table.	393
405	**405 ROT IN C AXIS** Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.	399
404	**404 SET BASIC ROTATION** Setting any basic rotation.	398

14.2 Fundamentals of the 14xx touch probe cycles

Characteristics common to the 14xx touch probe cycles for measuring rotations

Rotations can be determined using the following three cycles:
- 1410 **PROBING ON EDGE**
- 1411 **PROBING TWO CIRCLES**
- 1420 **PROBING IN PLANE**

These cycles include the following functionality:
- Consideration of active machine kinematics
- Semi-automatic probing
- Monitoring of tolerances
- Consideration of 3-D calibration
- Rotation and position are measured simultaneously

The programmed positions are interpreted as nominal positions in the I-CS. The probing positions reference the programmed nominal coordinates.

Evaluation – preset:
- If you want to probe objects in a consistent machining plane or probe on position objects while the TCPM is active, you can program any required shifts as basic transformations in the preset table.
- In the basic transformations, rotations can be programmed as basic rotations or as axial offsets from the first rotary table axis, seen from the workpiece.

Logging:

The determined results are recorded in the **TCHPRAUTO.html** file and stored in the Q-parameter programmed for this cycle.
The measured deviations refer to the mean value of the tolerance range. If you do not specify tolerances, they refer to the nominal dimension.
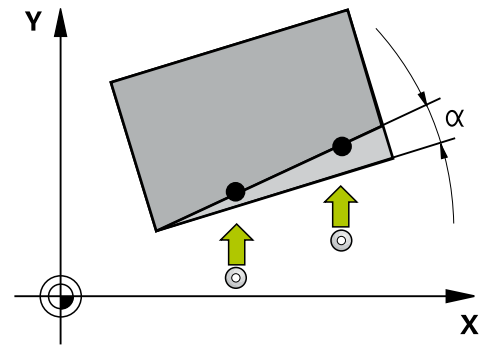
> 🛈 If you want to use not only the measured rotation, but also a measured position, make sure to probe the surface in its surface normal. The larger the angular error and the bigger the ball-tip radius, the larger the positioning error. If the angular errors in the initial angular position are too large, corresponding position errors might be the result.
>
> When probing with TCPM, existing 3-D calibration data are taken into account. If these calibration data do not exist, deviations might be the result.

Semi-automatic mode

If the workpiece placement has not been defined yet, you can use semi-automatic mode. In this mode, it is possible to determine the starting position by manually pre-positioning before you perform the probing operation on the desired object. This interruption is only possible in the machine operating modes, i.e. not in the Test run operating mode.

To do so, press the **ENTER TEXT** soft key and precede the nominal value of every coordinate of the object with a "**?**". If no nominal position has been defined, the control will perform an actual-to-nominal value transfer 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. This can be advantageously used to avoid corrections of the preset for directions that have not been defined accurately in a semi-automatic program run.

Cycle run:

- The cycle interrupts the normal program run.
- A dialog window opens.
- Use the axis-direction keys or the handwheel to pre-position the touch probe at the desired point.
- If required, change the probing conditions, such as the probing direction.
- Press **NC start**.**NC start**
- Make sure that the touch probe returns to a safe position at the end of the cycle to avoid any collision in the further program run.

NOTICE

Danger of collision!

Depending on the object to be probed, the control will ignore the programmed mode for retraction to clearance height when running in semi-automatic mode. If semi-automatic mode has only been programmed for a single object to be probed, the cycle will ignore the programmed retraction to clearance height for this object only.

▶ Make sure that the touch probe will return to a safe position at the end of the cycle.

Example:

When aligning the touch probe to an edge with 0° in Cycle 1410, presetting is to be performed in the reference axis direction, but not in the secondary and tool axes as these probing positions have not been defined accurately.

5 TCH PROBE 1410 PROBING TWO CIRCLES		Define the cycle
QS1100= "?10"	;1ST POINT REF AXIS	Nominal position 1 of reference axis exists, position of workpiece unknown
QS1101= "?"	;1ST POINT MINOR AXIS	Nominal position 1 of minor axis unknown
QS1102= "?"	;1ST POINT TOOL AXIS	Nominal position 1 of tool axis unknown
QS1103= "?50"	;2ND POINT REF AXIS	Nominal position 2 of reference axis exists, position of workpiece unknown
QS1104= "?"	;2ND POINT MINOR AXIS	Nominal position 2 of minor axis unknown
QS1105= "?"	;2ND POINT TOOL AXIS	Nominal position 2 of tool axis unknown
Q372=+1	;PROBING DIRECTION	Probing direction (-3 to +3)
...	;	

Evaluation of tolerances

Optionally, the control can monitor the touch probe system for tolerances, distinguishing between the position and dimension of an object.

If you added a tolerance to a dimension, the control will monitor it and set an error status in the **Q183** return parameter if the tolerance is not met. Tolerance monitoring and the status always refer to the situation during probing, i.e. before the preset is corrected by the cycle.

Cycle run:

- If reaction to errors has been enabled (**Q309=1**), the control will check for scrap and rework. If scrap is found, the execution of the NC program will be interrupted. If **Q309=2**, then the control will only check for scrap. If scrap is found, the execution of the NC program will be interrupted

- If your workpiece is considered as scrap, a dialog window opens. It shows all nominal and measured values of the object.

- You can then decide whether to continue machining or abort the program. To resume the program run, press **NC start**; to abort the program, press the **CANCEL** soft key.**CANCEL**

> **ⓘ** Please keep in mind that the touch probe cycles return the deviations with respect to the mean tolerance value in Q parameters **Q98x** and **Q99x**. Thus, these values equal the compensation values used by the cycle if input parameters **Q1120** and **Q1121** have been set accordingly. If no automatic evaluation has been programmed, it is thus easier to use these values, that reference the mean tolerance value, for other purposes.

5 TCH PROBE 1410 PROBING TWO CIRCLES		Define the cycle
Q1100=+50	;1ST POINT REF AXIS	Nominal position 1 of reference axis
Q1101= +50	;1ST POINT MINOR AXIS	Nominal position 1 of minor axis
Q1102= -5	;1ST POINT TOOL AXIS	Nominal position 1 of tool axis
QS1116="+9-1-0.5"	;DIAMETER 1	Diameter 1, tolerance specified
Q1103= +80	;2ND POINT REF AXIS	Nominal position 2 of reference axis
Q1104=+60	;2ND POINT MINOR AXIS	Nominal position 2 of minor axis
QS1105= -5	;2ND POINT TOOL AXIS	Nominal position 2 of tool axis
QS1117="+9-1-0,5"	;DIAMETER 2	Diameter 2, tolerance specified
...	;	
Q309=2	;ERROR REACTION	
...	;	

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. Based on the difference, the cycle calculates the required compensation values and applies tolerance monitoring.

Please note that in this case, the control does not perform probing, but only accounts for the actual and nominal positions.

To do so, press the **ENTER TEXT** soft key and follow the nominal value of each coordinate in the object with a "**@**". You can then enter the actual position after the "**@**".

> **i** You must define the actual position for all three axes: reference axis, minor 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 parameters **Q1900-Q1999**.

Example:

This feature allows you to do the following:

- Determine a circular pattern based on multiple different objects
- Align a gear wheel via its center and the position of a tooth

5 TCH PROBE 1410 PROBING ON EDGE	
QS1100= "10+0.02@10.0123"	
;1ST POINT REF AXIS	Nominal position 1 of the reference axis with tolerance monitoring and actual position
QS1101="50@50.0321"	
;1ST POINT MINOR AXIS	Nominal position 1 of the minor axis and actual position
QS1102= "-10-0.2+0.02@Q1900"	
;1ST POINT TOOL AXIS	Nominal position 1 of the tool axis with tolerance monitoring and actual position
... ;	

14.3 PROBING IN PLANE (Cycle 1420, ISO: G1420, software option 17)

Cycle run

Touch probe cycle 1420 finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

1 The control positions the touch probe at rapid traverse (value from the **FMAX** column), using positioning logic "Executing touch probe cycles", to the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.

2 The touch probe returns to the clearance height (depends on **Q1125**), then moves in the working plane to touch point **2** and measures the actual value of the second point of the plane.

3 The touch probe returns to the clearance height (depends on **Q1125**), then moves in the working plane to touch point **3** and measures the actual value of the third point of the plane.

4 Finally the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:
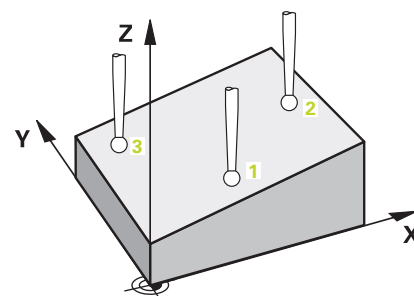
Parameter number	Meaning
Q950 to Q952	Measured position 1 in the principal, secondary, and tool axes
Q953 to Q955	Measured position 2 in the principal, secondary, and tool axes
Q956 to Q958	Measured position 3 in the principal, secondary, and tool axes
Q961 to Q963	Measured spatial angle SPA, SPB, and SPC in the WP-CS
Q980 to Q982	Measurement error 1 of the positions: principal, secondary, and tool axis
Q983 to Q985	Measurement error 2 of the positions: principal, secondary, and tool axis
Q986 to Q988	Measurement error 3 of the positions: principal, secondary, and tool axis
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

ℹ️ Before defining this cycle, you must have programmed a tool call to define the touch probe axis. This touch probe axis must be the Z axis.

The control can only calculate the angular values if the three measuring points are not positioned on a straight line.

Alignment with rotary axes is only possible if two rotary axes have been defined in the kinematics.

If **Q1121** equals 0 and **Q1126** is not equal to 0, an error message will be displayed because this would align the rotary axes, but the rotation would not be evaluated.

The deviations are the differences of the measured actual values to the mean tolerance value, not the difference to the nominal value.

The saved spatial angle is saved in parameters **Q961** to **Q963**. Based on the definition of the nominal positions, you can define the nominal spatial angle. The difference between the measured spatial angle and the nominal spatial angle is used to transfer the basic rotation in 3-D in the preset table.

Cycle parameters

- ▶ **Q1100 1st noml. position of ref. axis** (absolute): Nominal coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1101 1st noml. position of minor axis** (absolute): Nominal coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1102 1st nominal position tool axis?** (absolute): Nominal coordinate of the first touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1103 2nd noml. position of ref axis?** (absolute): Nominal coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1104 2nd noml. position of minor axis** (absolute): Nominal coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1105 2nd nominal pos. of tool axis?** (absolute): Nominal coordinate of the second touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1106 3rd noml. position of ref axis?** (absolute): Nominal coordinate of the third touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1107 3rd noml. position minor axis?** (absolute): Nominal coordinate of the third touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q1108 3rd nominal position tool axis?** (absolute): Nominal coordinate of the third touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input range: -3 to +3

- ▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | PROBING IN PLANE (Cycle 1420, ISO: G1420, software option 17)

14

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the measuring points:
-1: Do not move to clearance height
0: Move to clearance height before and after the cycle
1: Move to clearance height before and after moving to each measured object
2: Move to clearance height before and after moving to each measuring point

▶ **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt program run and display a message if a deviation is detected:
0: If the tolerance is exceeded, do not interrupt program run, do not display an error message
1: If the tolerance is exceeded, interrupt program run and display an error message
2: If the determined actual coordinate indicates that the workpiece is scrap, the control displays a message and interrupts program run. In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.

▶ **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
0: Maintain the current tilting axes position
1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
2: Position the tilting axis automatically without orienting the ball tip (TURN)

▶ **Q1120 Transfer position?**: Define which measured actual position is to be transfered to the preset table by the control:
0: Do not transfer any positions
1: Transfer measuring point 1
2: Transfer measuring point 2
3: Transfer measuring point 3
4: Transfer an averaged measuring point

▶ **Q1121 Confirm basic rotation?**: Define whether the control is to transfer the determined inclination as the basic rotation:
0: No basic rotation
1: Set basic rotation: The control saves the basic rotation.

Example

5 TCH PROBE 1420 ANTASTEN EBENE	
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

14.4 PROBING ON EDGE (Cycle 1410, ISO: G1410, software option 17)

Cycle run

Touch probe cycle 1410 measures the angle that any straight line on the workpiece forms with the reference axis of the working plane.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", to the programmed touch point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe in the direction opposite to the direction of probing.

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 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:
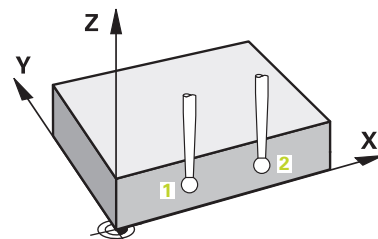
Parameter number	Meaning
Q950 to Q952	Measured position 1 in the principal, secondary, and tool axes
Q953 to Q955	Measured position 2 in the principal, secondary, and tool axes
Q964	Measured angle of rotation in the IP-CS
Q965	Measured angle of rotation in the coordinate system of the rotary table
Q980 to Q982	Measurement error 1 of the positions: principal, secondary, and tool axis
Q983 to Q985	Measurement error 2 of the positions: principal, secondary, and tool axis
Q994	Measured angle deviation in the IP-CS
Q995	Measured angle deviation in the coordinate system of the rotary table
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | PROBING ON EDGE (Cycle 1410, ISO: G1410, software option 17)

14

Please note while programming!

> **ⓘ** Before defining this cycle, you must have programmed a tool call to define the touch probe axis. This touch probe axis must be the Z axis.
>
> Alignment with rotary axes is only possible if the measured rotation can be compensated using a rotary table axis, which is the first rotary table axis, seen from the workpiece.
>
> If **Q1121** is not equal to 2 and **Q1126** is not equal to 0, an error message will be displayed. It would be contradictory to align the rotary axis and at the same time activate a basic rotation.
>
> The deviations are the differences of the measured actual values to the mean tolerance value (including the tolerance factor), not the difference to the nominal value.

Cycle parameters

▶ **Q1100 1st noml. position of ref. axis** (absolute): Nominal coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q1101 1st noml. position of minor axis** (absolute): Nominal coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q1102 1st nominal position tool axis?** (absolute): Nominal coordinate of the first touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q1103 2nd noml. position of ref axis?** (absolute): Nominal coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q1104 2nd noml. position of minor axis** (absolute): Nominal coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q1105 2nd nominal pos. of tool axis?** (absolute): Nominal coordinate of the second touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input range: -3 to +3

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | PROBING ON EDGE (Cycle 1410, ISO: G1410, software option 17)

14

▶ **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the measuring points:
-1: Do not move to clearance height
0: Move to clearance height before and after the cycle
1: Move to clearance height before and after moving to each measured object
2: Move to clearance height before and after moving to each measuring point

▶ **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt program run and display a message if a deviation is detected:
0: If the tolerance is exceeded, do not interrupt program run, do not display an error message
1: If the tolerance is exceeded, interrupt program run and display an error message
2: If the determined actual coordinate indicates that the workpiece is scrap, the control displays a message and interrupts program run. In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.

▶ **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
0: Maintain the current tilting axes position
1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
2: Position the tilting axis automatically without orienting the ball tip (TURN)

▶ **Q1120 Transfer position?**: Define which measured actual position is to be transfered to the preset table by the control:
0: Do not transfer any positions
1: Transfer measuring point 1
2: Transfer measuring point 2
3: Transfer an averaged measuring point

▶ **Q1121 CONFIRM ROTATION?**: Define whether the control is to transfer the determined inclination as the basic rotation:
0: No basic rotation
1: Set basic rotation: The control saves the basic rotation.
2: Perform rotation of the rotary table: The control will make an entry in the corresponding **Offset** column of the preset table

Example

5 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

14.5 PROBING TWO CIRCLES (Cycle 1411, ISO G1411, software option 17)

Cycle run

Touch probe cycle 1411 measures the centers of two holes or cylindrical studs. Then the control calculates the angle between the reference axis in the working plane and the line connecting the hole or 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.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic "Executing touch probe cycles", to the programmed center point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.

2 Then the probe moves to the entered measuring height and probes (depending on the number of probes in **Q423**) the first hole or stud center point.

3 The touch probe returns to the clearance height and then to the position entered as center of the second hole or second stud **2**.

4 Then the control moves the probe to the entered measuring height and probes (depending on the number of probes in **Q423**) the second hole or stud center point.

5 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:
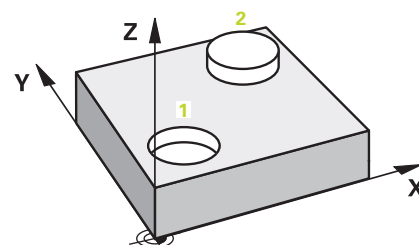
Parameter number	Meaning
Q950 to Q952	Measured position 1 in the principal, secondary, and tool axes
Q953 to Q955	Measured position 2 in the principal, secondary, and tool axes
Q964	Measured angle of rotation in the IP-CS
Q965	Measured angle of rotation in the coordinate system of the rotary table
Q966 to Q967	Measured first and second diameters
Q980 to Q982	Measurement error 1 of the positions: principal, secondary, and tool axis
Q983 to Q985	Measurement error 2 of the positions: principal, secondary, and tool axis
Q994	Measured angle deviation in the IP-CS

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | PROBING TWO CIRCLES (Cycle 1411, ISO G1411, software option 17)

14

Parameter number	Meaning
Q995	Measured angle deviation in the coordinate system of the rotary table
Q996 to Q997	Measurement error of the first and second diameters
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

ℹ Before defining this cycle, you must have programmed a tool call to define the touch probe axis. This touch probe axis must be the Z axis.

Alignment with rotary axes is only possible if the measured rotation can be compensated using a rotary table axis, which is the first rotary table axis, seen from the workpiece.

If **Q1121** is not equal to 2 and **Q1126** is not equal to 0, an error message will be displayed. It would be contradictory to align the rotary axis and at the same time activate a basic rotation.

The deviations are the differences of the measured actual values to the mean tolerance value, not the difference to the nominal value.

If the hole diameter is less than the ball tip diameter, an error message will be displayed.

Is the hole diameter is so small that the programmed set-up clearance cannot be achieved, a dialog opens. The dialog shows the nominal value corresponding to the hole radius, the calibrated ball-tip radius and the achievable set-up clearance. Either confirm the dialog with **NC start** or cancel the process via a soft key. If you confirm with **NC start**, the effective set-up clearance will only be reduced to the displayed value, but only for this object to be probed.
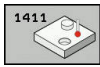
Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | PROBING TWO CIRCLES
(Cycle 1411, ISO G1411, software option 17)

14

Cycle parameters

- **Q1100 1st noml. position of ref. axis** (absolute): Nominal coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Q1101 1st noml. position of minor axis** (absolute): Nominal coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Q1102 1st nominal position tool axis?** (absolute): Nominal coordinate of the first touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Q1116 Diameter of 1st position?**: Diameter of the first hole or stud. Input range: 0 to 9999.9999

- **Q1103 2nd noml. position of ref axis?** (absolute): Nominal coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Q1104 2nd noml. position of minor axis** (absolute): Nominal coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Q1105 2nd nominal pos. of tool axis?** (absolute): Nominal coordinate of the second touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Q1117 Diameter of 2nd position?**: Diameter of the second hole or stud. Input range: 0 to 9999.9999

- **Q1115 Geometry type (0-3)?**: Define the geometry of the objects 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

- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8

- **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000
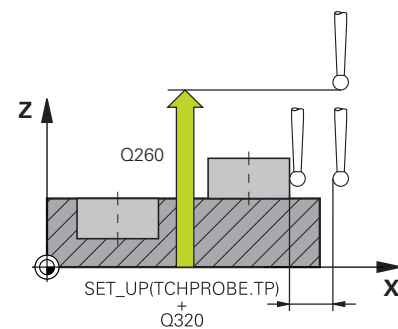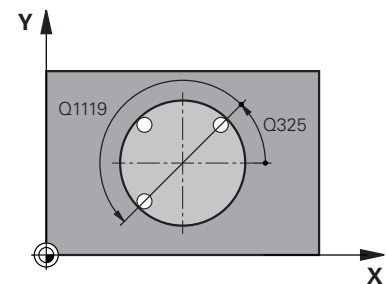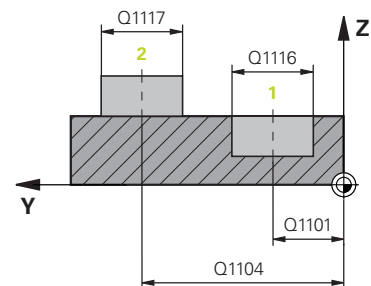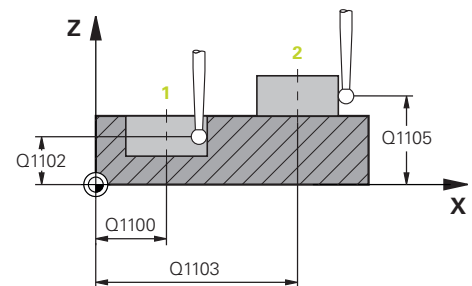
▶ **Q1119 Arc angular length?**: Angular range in which the touch points are distributed. Input range: -359.999 to +360

▶ **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the measuring points:
-1: Do not move to clearance height
0: Move to clearance height before and after the cycle
1: Move to clearance height before and after moving to each measured object
2: Move to clearance height before and after moving to each measuring point

▶ **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt program run and display a message if a deviation is detected:
0: If the tolerance is exceeded, do not interrupt program run, do not display an error message
1: If the tolerance is exceeded, interrupt program run and display an error message
2: If the determined actual coordinate indicates that the workpiece is scrap, the control displays a message and interrupts program run. In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.

▶ **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
0: Maintain the current tilting axes position
1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
2: Position the tilting axis automatically without orienting the ball tip (TURN)

▶ **Q1120 Transfer position?**: Define which measured actual position is to be transfered to the preset table by the control:
0: Do not transfer any positions
1: Transfer measuring point 1
2: Transfer measuring point 2
3: Transfer an averaged measuring point

Example

5 TCH PROBE 1410 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

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | PROBING TWO CIRCLES (Cycle 1411, ISO G1411, software option 17)

14

▶ **Q1121 CONFIRM ROTATION?**: Define whether the control is to transfer the determined inclination as the basic rotation:
0: No basic rotation
1: Set basic rotation: The control saves the basic rotation.
2: Perform rotation of the rotary table: The control will make an entry in the corresponding **Offset** column of the preset table

14.6 Fundamentals of the 4xx touch probe cycles

Characteristics common to all touch probe cycles for measuring workpiece misalignment

For Cycles 400, 401 and 402 you can define through parameter **Q307 Preset value for rotation angle** whether the measurement result is to be corrected by a known angle # (see figure at right). 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**.

ℹ️ These cycles do not work with 3-D Rot! In such a case, use Cycles 14xx. **Further information:** "Fundamentals of the 14xx touch probe cycles ", Page 361

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | BASIC ROTATION (Cycle 400, ISO: G400, software option 17)

14
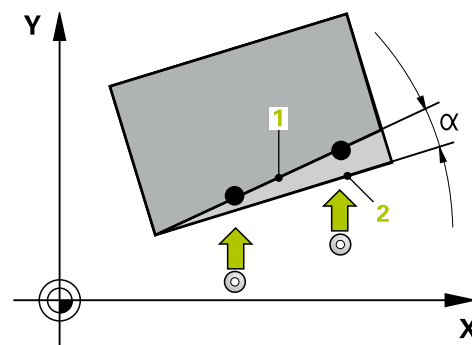
14.7 BASIC ROTATION (Cycle 400, ISO: G400, software option 17)

Cycle run

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.

1   The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), 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.

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.

Please note while programming:

> ℹ️   Before defining this cycle, 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.

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶   The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶   Reset any coordinate transformations beforehand

Cycle parameters

- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis

- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction

- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

- ▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points

Example

5 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

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | BASIC ROTATION (Cycle 400, ISO: G400, software option 17)

14

▶ **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the reference axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: -360.000 to 360.000

▶ **Q305 Preset number in table?**: Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. If you enter Q305=0, the control automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. Input range: 0 to 99999

14.8 BASIC ROTATION over two holes (Cycle 401, ISO: G401, software option 17)

Cycle run

Touch probe cycle 401 measures the center points of two holes. The control then calculates the angle between the reference axis of the working plane and the line connecting the hole 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.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the programmed center point of the first hole **1**.

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.

Please note while programming:

> **ℹ** Before defining this cycle, 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.
>
> 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

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

Cycle parameters

▶ **Q268 1st hole: center in 1st axis?** (absolute):
Center of the first hole in the reference axis of
the working plane. Input range: -99999.9999 to
99999.9999

▶ **Q269 1st hole: center in 2nd axis?** (absolute):
Center of the first hole in the minor axis of the
working plane. Input range: -99999.9999 to
99999.9999

▶ **Q270 2nd hole: center in 1st axis?** (absolute):
Center of the second hole in the reference axis
of the working plane. Input range: -99999.9999 to
99999.9999

▶ **Q271 2nd hole: center in 2nd axis?** (absolute):
Center of the second hole in the minor axis of
the working plane. Input range: -99999.9999 to
99999.9999

▶ **Q261 Measuring height in probe axis?** (absolute):
Coordinate of the ball tip center (= touch point) in
the touch probe axis in which the measurement
is to be made. Input range: -99999.9999 to
99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in
the touch probe axis at which no collision between
tool and workpiece (fixtures) can occur. Input
range: -99999.9999 to 99999.9999

Example

5 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

▶ **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the reference axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: -360.000 to 360.000

▶ **Q305 Number in table?** Enter the number of a row in the preset table. The control will enter the value in this row: Input range: 0 to 99999
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, an entry is made in column **SPC**)
Q337 = 0 and, at the same time, **Q402** = 1: Parameter Q305 is not effective
Q337 = 1 Parameter Q305 is in effect as described above

▶ **Q402 Basic rotation/alignment (0/1)**: Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of 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 **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

▶ **Q337 Set to zero after alignment?**: Define whether the control is to 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 defined **Q402=1**

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

14.9 BASIC ROTATION over two studs (Cycle 402, ISO: G402, software option 17)

Cycle run

Touch probe cycle 402 measures the center points of two cylindrical studs. The control then calculates the angle between the reference 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.

1 The control positions the touch probe at rapid traverse speed (value from FMAX column), using positioning logic (see "Executing touch probe cycles", Page 355), to the programmed center point of the first stud **1**.

2 Then the probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves on 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 positions the probe to starting 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 second stud center point.

5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.

Please note while programming:

> **ℹ** Before defining this cycle, 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.
>
> 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

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
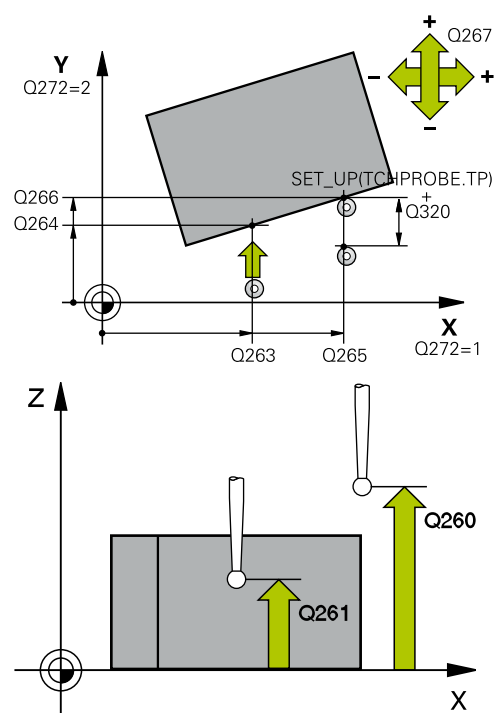
▶ Reset any coordinate transformations beforehand

Cycle parameters

▶ **Q268 1st stud: center in 1st axis?** (absolute): Center of the first stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q269 1st stud: center in 2nd axis?** (absolute): Center of the first stud in the minor axis of the working plane. Input range: -99999.9999 to 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 range: 0 to 99999.9999

▶ **Q261 Meas. height stud 1 in TS axis?** (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 1 is to be measured. Input range: -99999.9999 to 99999.9999

▶ **Q270 2nd stud: center in 1st axis?** (absolute): Center of the second stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q271 2nd stud: center in 2nd axis?** (absolute): Center of the second stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q313 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 range: 0 to 99999.9999

▶ **Q315 Meas. height stud 2 in TS axis?** (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 2 is to be measured. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points
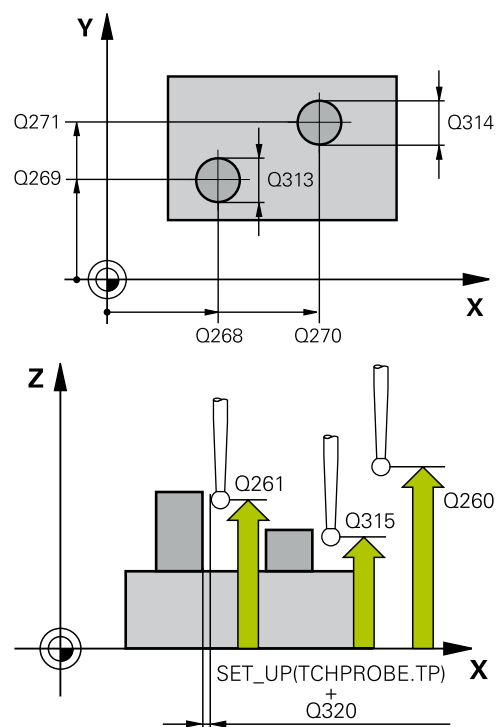
Example

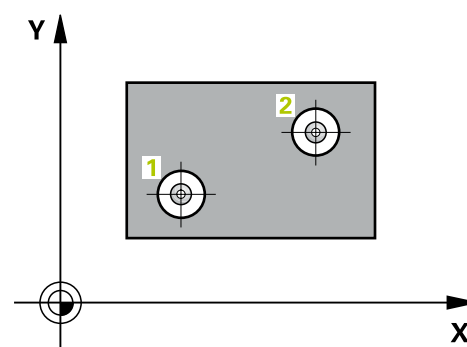| 5 TCH PROBE 402 ROT OF 2 STUDS	
Q268=-37	;1ST CENTER 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q313=60	;DIAMETER OF STUD 1
Q261=-5	;MEAS. HEIGHT STUD 1
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q314=60	;DIAMETER OF STUD 2
Q315=-5	;MEAS. HEIGHT STUD 2
Q320=0	;SET-UP CLEARANCE
Q260=+20	;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

▶ **Q307 Preset value for rotation angle** (absolute):
If the misalignment is to be measured against
any straight line rather than the reference axis,
enter the angle of this reference line. The control
will then calculate the difference between the
value measured and the angle of the reference
line for the basic rotation. Input range: -360.000 to
360.000

▶ **Q305 Number in table?** Enter the number of a
row in the preset table. The control will enter the
value in this row: Input range: 0 to 99999
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, an entry is made
in column **SPC**)
Q337 = 0 and, at the same time, **Q402** = 1:
Parameter Q305 is not effective
Q337 = 1 Parameter Q305 is in effect as described
above

▶ **Q402 Basic rotation/alignment (0/1)**: Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of 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 **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

▶ **Q337 Set to zero after alignment?**: Define whether the control is to 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 defined **Q402=1**

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | BASIC ROTATION compensation via rotary axis (Cycle 403, ISO: G403, software option 17)

14

14.10 BASIC ROTATION compensation via rotary axis (Cycle 403, ISO: G403, software option 17)

Cycle run

Touch probe cycle 403 determines a workpiece misalignment by measuring two points, which must lie on a straight line. The control compensates the determined misalignment by rotating the A, B, or C axis. The workpiece can be clamped in any position on the rotary table.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), 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.

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.
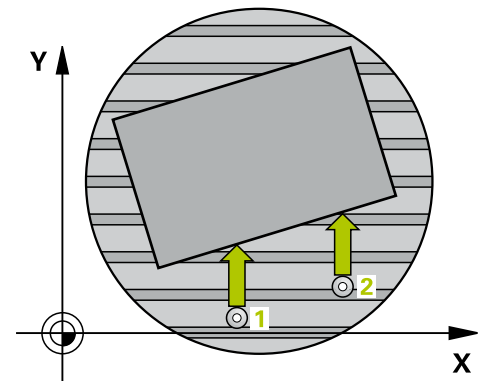
Please note while programming:

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 lies in the range from -90° to +90°.

▶ After alignment, check the position of the rotary axis.

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis
3: Touch probe axis = measuring axis

▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
–1: Negative traverse direction
+1: Positive traverse direction

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

Example

5 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

▶ **Q312 Axis for compensating movement?**:
Specify the rotary axis in which the control is to 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

▶ **Q337 Set to zero after alignment?**: Define whether the control should 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 alignment
1: Set the angle of the rotary axis to 0 in the table after alignment

▶ **Q305 Number in table?** Specify the number of the row in the preset table in which the control is to enter the basic rotation. Input range: 0 to 99999
Q305 = 0: The rotary axis is zeroed in row 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 is to 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 is effective as described above
Q312 = 0: Parameter Q305 is effective as 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.

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q380 Ref. angle in ref. axis?**: Angle to which the control is to 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 range: -360.000 to 360.000

14.11 SET BASIC ROTATION (Cycle 404, ISO: G404, software option 17)

Cycle run

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.

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

Cycle parameters

▶ **Q307 Preset value for rotation angle**: Angular value to which the basic rotation is to be set. Input range: -360.000 to 360.000

▶ **Q305 Preset number in table?**: Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. Input range: -1 to 99999. If you enter Q305=0 or Q305=-1, the control additionally saves the determined basic rotation in the basic rotation menu (**Probing rot**) of the **Manual Operation** mode.
-1 = Overwrite and activate the active the active preset
0 = Copy the active preset to preset row 0, write basic rotation to preset row 0 and activate preset 0
>1 = Save the basic rotation to the specified preset. The preset is not activated.

Example

5 TCH PROBE 404 SET BASIC ROTATION	
Q307=+0	;PRESET ROTATION ANG.
Q305=-1	;NUMBER IN TABLE

14.12 Compensating workpiece misalignment by rotating the C axis (Cycle 405, ISO: G405, software option 17)

Cycle run

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, or
- the angular offset between the nominal position and the actual position of a hole center.

The control compensates 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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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 and then positions the touch probe on the hole center measured.

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—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in parameter Q150.
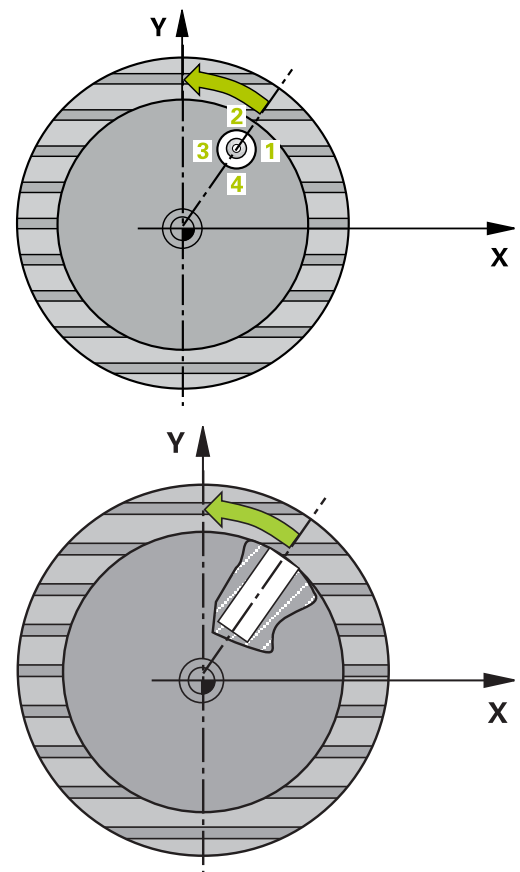
Please note while programming:

> ℹ️ ▶ Before a cycle definition you must have programmed
> a tool call to define the touch probe axis.
>
> ▶ The smaller the stepping angle, the less accurately
> the control can calculate the circle center point.
> Minimum input value: 5°

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.

▶ Make sure that there is no material in the pocket/hole

▶ To prevent a collision between the touch probe and the
workpiece, enter a **low** estimate for the nominal diameter of
the pocket (or hole).

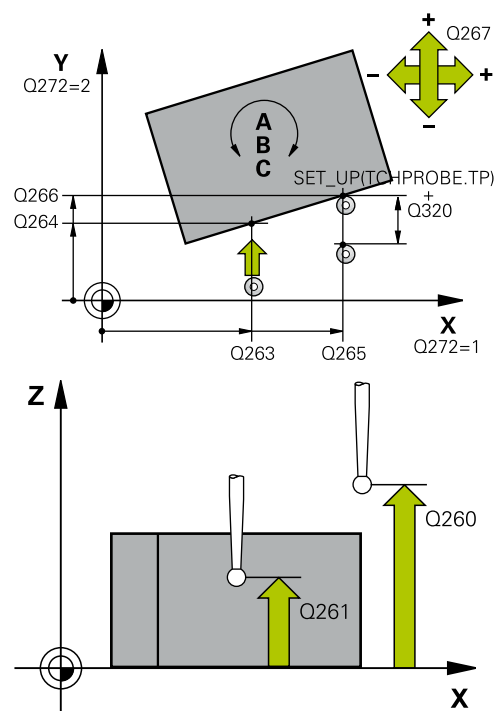---

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for
coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch
probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**,
Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-
SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | Compensating workpiece misalignment by rotating the C axis (Cycle 405, ISO: G405, software option 17)

14

Cycle parameters

- ▶ **Q321 Center in 1st axis?** (absolute): Center of the hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the hole in the minor 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 # 0, then the control aligns the hole center point with the nominal position (angle resulting from the position of the hole center). Input range: -99999.9999 to 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 range: 0 to 99999.9999

- ▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000

- ▶ **Q247 Intermediate stepping angle?** Q247 (incremental): 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°. Input range: -120.000 to 120.000

- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

- ▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

Example

5 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

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q337 Set to zero after alignment?**:
0: Set the display of the C axis to 0 and write value 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 registered in the datum table, the control adds the measured angular offset with the correct sign, positive or negative.

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment | Example: Determining a basic rotation from two holes

14

14.13 Example: Determining a basic rotation from two holes

0 BEGIN P GM CYC401 MM		
1 TOOL CALL 69 Z		
2 TCH PROBE 401 ROT OF 2 HOLES		
Q268=+25	;1ST CENTER 1ST AXIS	Center of the 1st hole: X coordinate
Q269=+15	;1ST CENTER 2ND AXIS	Center of the 1st hole: Y coordinate
Q270=+80	;2ND CENTER 1ST AXIS	Center of the 2nd hole: X coordinate
Q271=+35	;2ND CENTER 2ND AXIS	Center of the 2nd hole: Y coordinate
Q261=-5	;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+20	;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q307=+0	;PRESET ROTATION ANG.	Angle of the reference line
Q305=0	;NUMBER IN TABLE	
Q402=1	;COMPENSATION	Compensate misalignment by rotating the rotary table
Q337=1	;SET TO ZERO	Set the display to zero after the alignment
3 CALL PGM 35K47		Call part program
4 END PGM CYC401 MM		

15

Touch Probe Cycles: Automatic Presetting

15.1 Fundamentals

Overview

The control offers twelve cycles for automatically finding presets and handling them as follows:

- Setting the determined values directly as display values
- Writing the determined values to the preset table
- Writing the determined values to a datum table

Soft key	Cycle	Page
408	408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as preset	409
409	409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as preset	413
410	410 DATUM INSIDE RECTAN-GLE Measuring the inside length and width of a rectangle, and defining the center as preset	417
411	411 DATUM OUTSIDE RECTANGLE Measuring the outside length and width of a rectangle, and defining the center as preset	421
412	412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as preset	425
413	413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as preset	430
414	414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as preset	435
415	415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as preset	440
416	416 DATUM CIRCLE CENTER (2nd soft-key level) Measuring any three holes on a bolt hole circle, and defining the bolt-hole center as preset	445

Soft key	Cycle	Page
417	417 PRESET IN TS AXIS (2nd soft-key row) Measuring any position in the touch probe axis and defining it as preset	450
418	418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as preset	452
419	419 DATUM IN ONE AXIS (2nd soft-key row) Measuring any position in any axis and defining it as preset	457

⊙ The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

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.

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
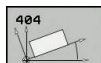
▶ Reset any coordinate transformations beforehand

Characteristics common to all touch probe cycles for presetting

> **i** You can also run the Touch Probe Cycles 408 to 419 during an active rotation (basic rotation or Cycle 10).

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 the 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 and activates row 0. This will delete simple transformations.

- **Q305 not equal to 0, Q303 = 0:**
  The result is written to the datum table, row **Q305**. **Activate the datum with Cycle 7 in the NC program.**

- **Q305 not equal to 0, Q303 = 1:**
  The control writes the result to the preset table, row **Q305**. The reference system is the machine coordinate system (REF coordinates). **Activate the preset with Cycle 247 in the NC program.**

- **Q305 not equal to 0, Q303 = -1**

> **i** This combination can only occur if you
> - read in NC programs containing Cycles 410 to 418 created on a TNC 4xx
> - read in NC programs containing Cycles 410 to 418 created with an older software version on 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.

15.2 SLOT CENTER PRESET (Cycle 408, ISO: G408, software option 17)

Cycle run

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.

1   The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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   Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and saves the actual values in the Q parameters listed below.

5   If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Please note while programming:
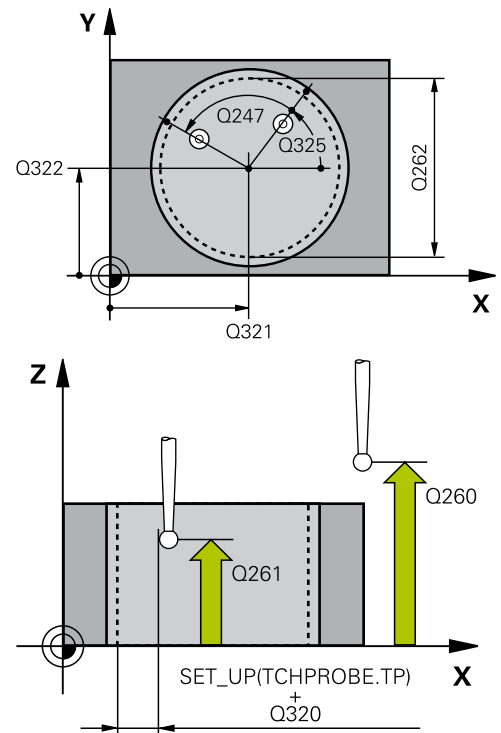

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width. 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.

▶ Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters
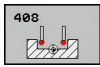
▶ **Q321 Center in 1st axis?** (absolute): Center of the slot in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q322 Center in 2nd axis?** (absolute): Center of the slot in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q311 Width of slot?** (incremental): Width of the slot, regardless of its position in the working plane. Input range: 0 to 99999.9999

▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
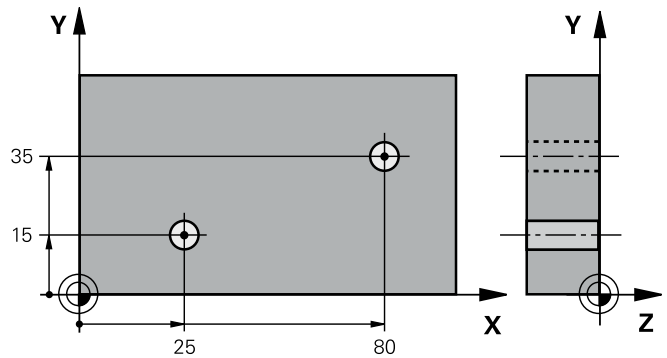
▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated slot center. Default setting = 0. Input range: -99999.9999 to 99999.9999

SET_UP(TCHPROBE.TP)
+
Q320
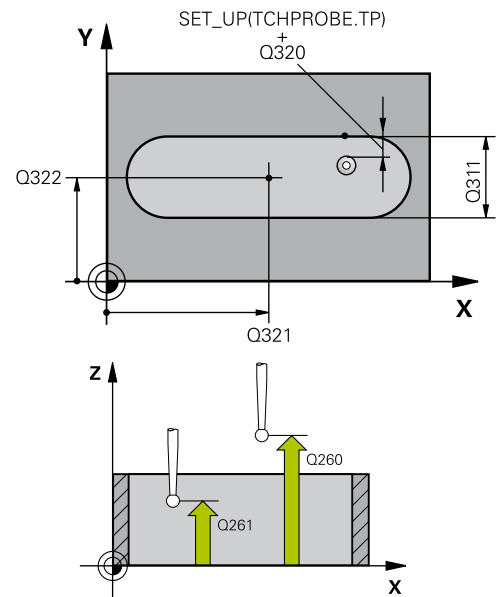
Example

5 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

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

15.3 RIDGE CENTER PRESET (Cycle 409, ISO: G409, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and saves the actual values in the Q parameters listed below.

5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.
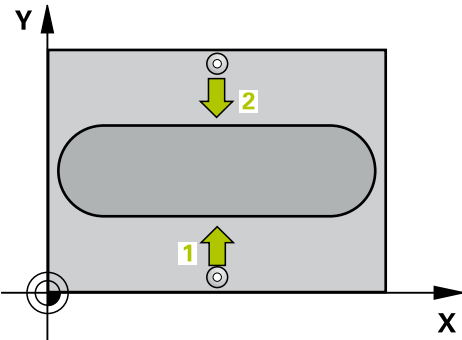
Parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

Please note while programming:

NOTICE
Danger of collision!
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **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.

Cycle parameters

▶ **Q321 Center in 1st axis?** (absolute): Center of the ridge in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q322 Center in 2nd axis?** (absolute): Center of the ridge in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q311 Ridge width?** (incremental): Width of the ridge, regardless of its position in the working plane. Input range: 0 to 99999.9999

▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

Example

5 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

▶ **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated ridge center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET

Touch Probe Cycles: Automatic Presetting | PRESET FROM INSIDE OF RECTANGLE (Cycle 410, ISO: G410, software option 17)

15

15.4 PRESET FROM INSIDE OF RECTANGLE (Cycle 410, ISO: G410, software option 17)

Cycle run

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.

1   The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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 clearance height and processes the determined preset depending on cycle parameters Q303 and Q305. (see "Characteristics common to all touch probe cycles for presetting", Page 408)

6   If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.

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

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand


NOTICE

Danger of collision!

To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides. 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.

▶ Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters

▶ **Q321 Center in 1st axis?** (absolute): Center of the pocket in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q323 First side length?** (incremental): Pocket length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q324 Second side length?** (incremental): Pocket length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999

Example

| 5 TCH PROBE 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

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

15.5 PRESET FROM OUTSIDE OF RECTANGLE (Cycle 411, ISO: G411, software option 17)

Cycle run

Touch probe cycle 411 finds the center of a rectangular 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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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 clearance height and processes the determined preset depending on cycle parameters Q303 and Q305. (see "Characteristics common to all touch probe cycles for presetting", Page 408)

6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.

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

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **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 a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters

▶ **Q321 Center in 1st axis?** (absolute): Center of the stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q322 Center in 2nd axis?** (absolute): Center of the stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q323 First side length?** (incremental): Stud length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q324 Second side length?** (incremental): Stud length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999
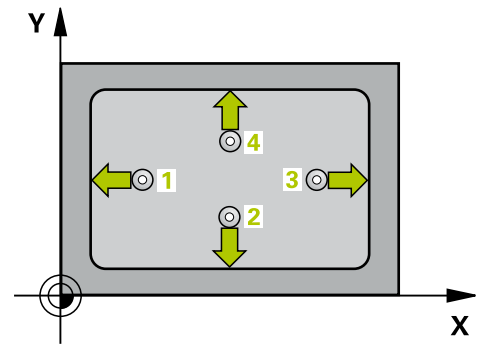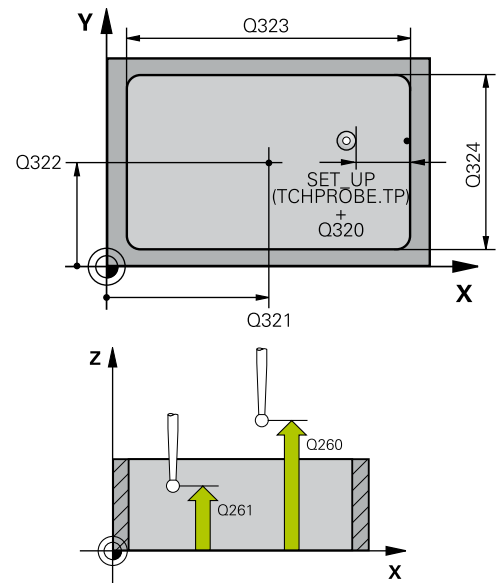
Example

5 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

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
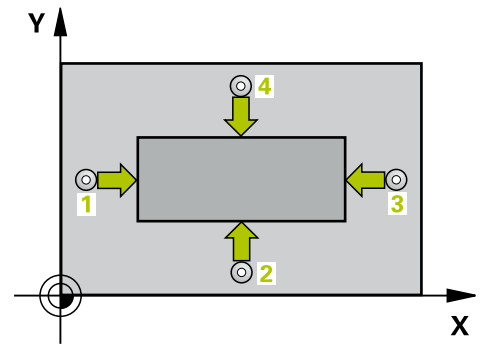
▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

15.6 PRESET FROM INSIDE OF CIRCLE (Cycle 412, ISO: G412, software option 17)

Cycle run

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.

1   The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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 processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and saves the actual values in the Q parameters listed below.

6   If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

> **ⓘ** ▶ The smaller the stepping angle Q247, the less accurately the control can calculate the preset. Minimum input value: 5°
>
> ▶ Program a stepping angle to be less than 90°; input range -120° to 120°

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole). 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.

▶ Positioning of the touch points

▶ Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters

▶ **Q321 Center in 1st axis?** (absolute): Center of the pocket in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the minor 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. Input range: -99999.9999 to 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 range: 0 to 99999.9999

▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000

▶ **Q247 Intermediate stepping angle?** Q247 (incremental): 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°. Input range: -120.000 to 120.000

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

Example

5 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

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

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**

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
4: Use 4 measuring points (default setting)
3: Use 3 measuring points

▶ **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (Q301=1):
0: Move in a straight line between machining operations
1: Move in a circular arc on the pitch circle diameter between machining operations

15.7 PRESET FROM OUTSIDE OF CIRCLE (Cycle 413, ISO: G413, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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 processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and saves the actual values in the Q parameters listed below.

6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

> 🛈
> ▶ The smaller the stepping angle Q247, the less accurately the control can calculate the preset. Minimum input value: 5°
> ▶ Program a stepping angle to be less than 90°; input range -120° to 120°

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand
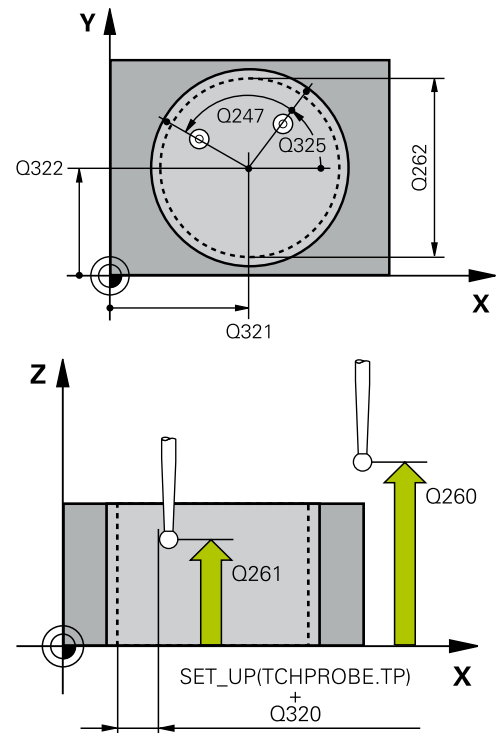

NOTICE

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.

Cycle parameters

▶ **Q321 Center in 1st axis?** (absolute): Center of the stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q322 Center in 2nd axis?** (absolute): Center of the stud in the minor 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. Input range: -99999.9999 to 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 range: 0 to 99999.9999

▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000

▶ **Q247 Intermediate stepping angle?** Q247 (incremental): 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°. Input range: -120.000 to 120.000

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

Example

5 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

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
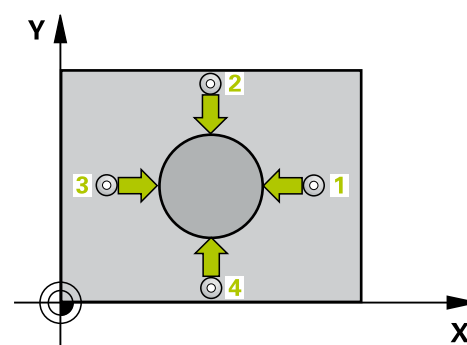
▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
0: Do not set the preset in the touch probe axis
1: Set the preset in the touch probe axis

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
4: Use 4 measuring points (default setting)
3: Use 3 measuring points

▶ **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (Q301=1):
0: Move in a straight line between machining operations
1: Move in a circular arc on the pitch circle diameter between machining operations

15.8 PRESET FROM OUTSIDE OF CORNER (Cycle 414, ISO: G414, software option 17)

Cycle run

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.

1   The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1** (see figure at right). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.

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   Finally the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and saves the coordinates of the determined corner in the Q parameters listed below.

6   If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:

<table>
<tr><td colspan="2" align="center">*NOTICE*</td></tr>
<tr><td colspan="2">**Danger of collision!**

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

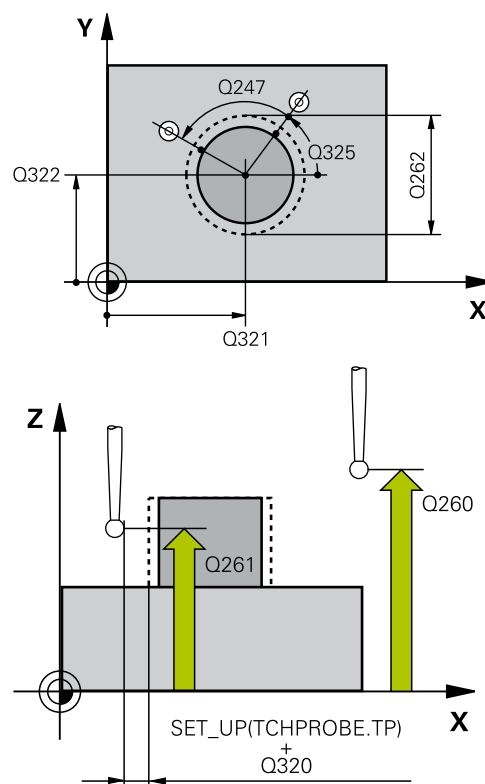▶ Reset any coordinate transformations beforehand</td></tr>
</table>

i Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control always measures the first line in the direction of the minor axis of the working plane.

By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see figure at the right and table below).
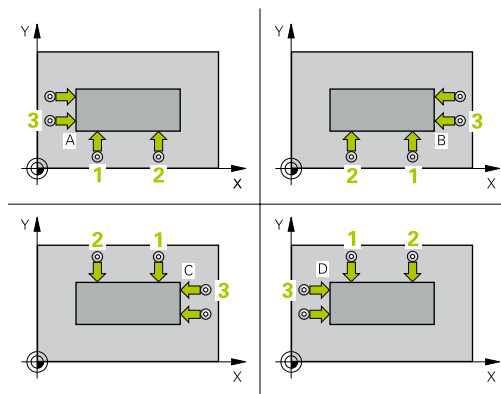
Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

Example

5 TCH PROBE 414 PRESET INSIDE 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

▶ **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:
0: Do not execute basic rotation
1: Execute basic rotation

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the corner coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

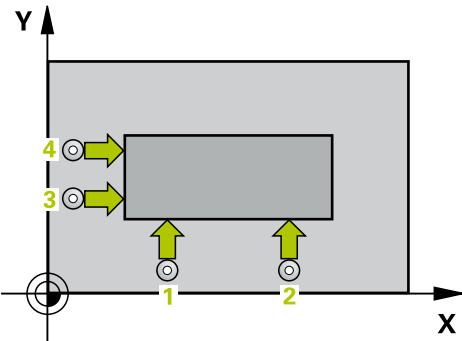|	
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

15.9 PRESET FROM INSIDE OF CORNER (Cycle 415, ISO: G415, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1** defined in the cycle (see figure at right). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.

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 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 Finally the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and saves the coordinates of the determined corner in the Q parameters listed below.

6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:

| *NOTICE* |
|---|
| **Danger of collision!** |
| When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active. |
| ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING** |
| ▶ Reset any coordinate transformations beforehand |

ℹ Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control always measures the first line in the direction of the minor axis of the working plane.

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q308 Corner? (1/2/3/4)**: Number identifying the corner which the control is to set as the preset. Input range: 1 to 4

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:
0: Do not execute basic rotation
1: Execute basic rotation
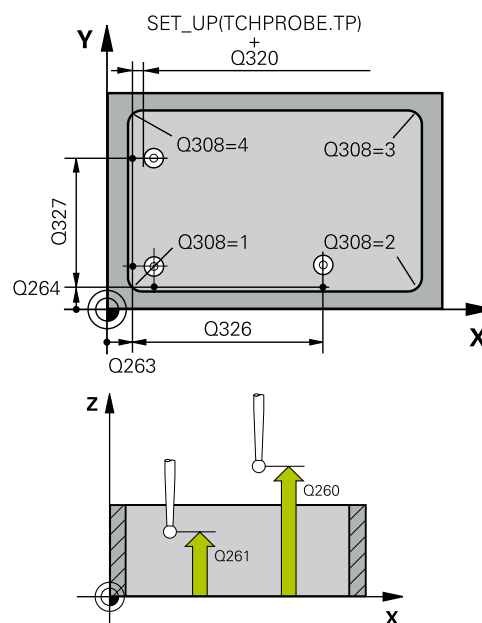
Example

5 TCH PROBE 415 PRESET OUTS. 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	

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the corner coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
0: Do not set the preset in the touch probe axis
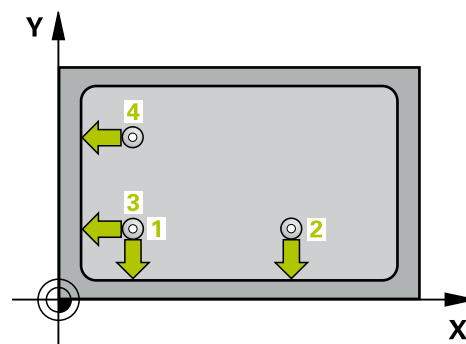**1**: Set the preset in the touch probe axis

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):
Coordinate of the probe point in the reference axis
of the working plane at which point the preset is
to be set in the touch probe axis. Only effective if
Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):
Coordinate of the probe point in the minor axis
of the working plane at which point the preset is
to be set in the touch probe axis. Only effective if
Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):
Coordinate of the touch point in the touch probe
axis, at which point the preset is to be set in the
touch probe axis. Only effective if Q381 = 1. Input
range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute):
Coordinate in the touch probe axis at which the
control should set the preset. Default setting = 0.
Input range: -99999.9999 to 99999.9999

15.10 PRESET CIRCLE CENTER (Cycle 416, ISO: G416, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the programmed center point of the first hole **1**.

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 processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and 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.
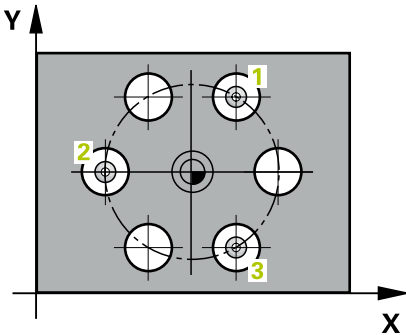
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**

▶ Reset any coordinate transformations beforehand

ⓘ Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters
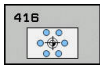
▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range: -99999.9999 to 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 range: -0 to 99999.9999

▶ **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range: -360.0000 to 360.0000

▶ **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range: -360.0000 to 360.0000

▶ **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range: -360.0000 to 360.0000

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the bolt-hole circle center. Default setting = 0. Input range: -99999.9999 to 99999.9999

Example

5 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

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the bolt-hole circle center. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999

15.11 PRESET IN TOUCH PROBE AXIS (Cycle 417, ISO: G417, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the positive direction of the touch probe axis.

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 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408) and saves the actual values in the Q parameters listed below.
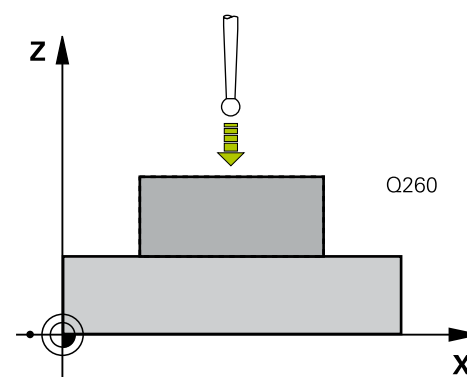
Parameter number	Meaning
Q160	Actual value of measured point

### Please note while programming:

> **NOTICE**

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
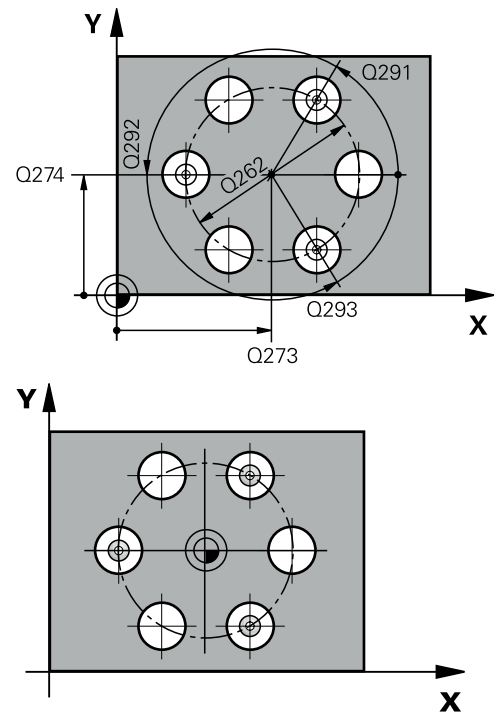
▶ Reset any coordinate transformations beforehand

ℹ️ Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control then sets the preset in this axis.

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates; input range: 0 to 9999.
If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q333 New preset in TS axis?** (absolute): Coordinate at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

Example

5 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

15.12 PRESET AT CENTER OF 4 HOLES (Cycle 418, ISO: G418, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the center point of the first hole **1**.

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 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 408). 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.

7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.
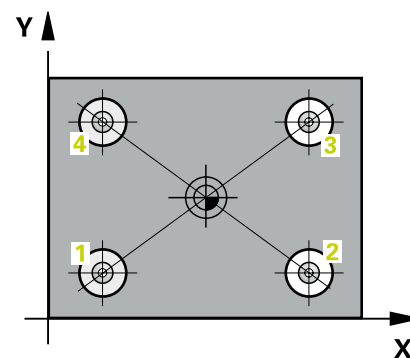
Parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

Please note while programming:

NOTICE
Danger of collision!
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
▶ Reset any coordinate transformations beforehand

i	Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

▶ **Q268 1st hole: center in 1st axis?** (absolute): Center of the first hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q269 1st hole: center in 2nd axis?** (absolute): Center of the first hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q270 2nd hole: center in 1st axis?** (absolute): Center of the second hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q271 2nd hole: center in 2nd axis?** (absolute): Center of the second hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q316 3rd hole: Center in 1st axis?** (absolute): Center of the third hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q317 3rd hole: Center in 2nd axis?** (absolute): Center of the third hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q318 4th hole: Center in 1st axis?** (absolute): Center of the fourth hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q319 4th hole: Center in 2nd axis?** (absolute): Center of the fourth hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

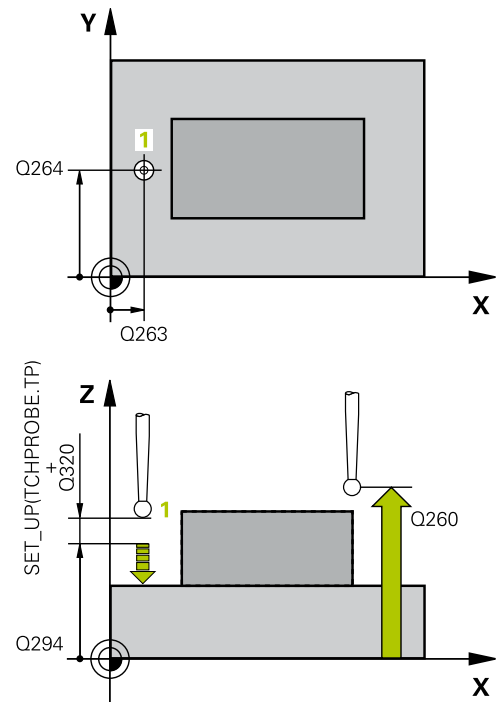▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
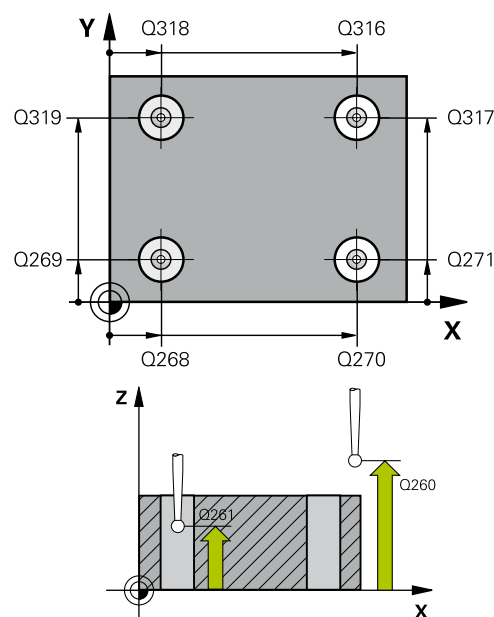
Example

5 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

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates of the point of intersection of the connecting lines; input range: 0 to 9999.
If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the intersection of the connecting lines. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the intersection of the connecting lines. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should 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

▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

15.13 PRESET IN ONE AXIS (Cycle 419, ISO: G419, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite to the programmed direction of probing.

2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.

3 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters Q303 and Q305. (see "Characteristics common to all touch probe cycles for presetting", Page 408)

Please note while programming:

> ### *NOTICE*
>
> **Danger of collision!**
>
> When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
>
> ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
>
> ▶ Reset any coordinate transformations beforehand

> **ℹ** Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> 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.

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis
3: Touch probe axis = measuring axis

Axis assignment

Active touch probe axis: Q272 = 3	Associated reference axis: Q272 = 1	Associated minor axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
–1: Negative traverse direction
+1: Positive traverse direction

Example

5 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

▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates; input range: 0 to 9999.
If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.

▶ **Q333 New preset in TS axis?** (absolute): Coordinate at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to 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 read in (see "Characteristics common to all touch probe cycles for presetting", Page 408)
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

15.14 Example: Presetting in center of a circular segment and on top surface of workpiece
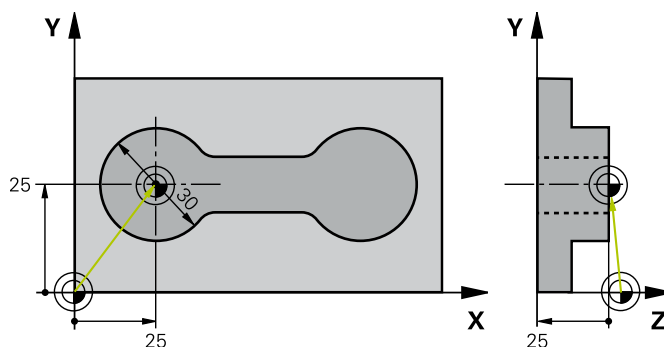
0 BEGIN PGM CYC413 MM		
1 TOOL CALL 69 Z		
2 TCH PROBE 413 PRESET OUTS. CIRCLE		
Q321=+25	;CENTER IN 1ST AXIS	Center of circle: X coordinate
Q322=+25	;CENTER IN 2ND AXIS	Center of circle: Y coordinate
Q262=30	;NOMINAL DIAMETER	Circle diameter
Q325=+90	;STARTING ANGLE	Polar coordinate angle for 1st touch point
Q247=+45	;STEPPING ANGLE	Stepping angle for calculating the starting points 2 to 4
Q261=-5	;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q320=2	;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+10	;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q301=0	;MOVE TO CLEARANCE	Do not move to clearance height between measuring points
Q305=0	;NUMBER IN TABLE	Set display
Q331=+0	;PRESET	Set the display in X to 0
Q332=+10	;PRESET	Set the display in Y to 10
Q303=+0	;MEAS. VALUE TRANSFER	Without function, since display is to be set
Q381=1	;PROBE IN TS AXIS	Also set preset in the touch probe axis
Q382=+25	;1ST CO. FOR TS AXIS	X coordinate of touch point
Q383=+25	;2ND CO. FOR TS AXIS	Y coordinate of touch point
Q384=+25	;3RD CO. FOR TS AXIS	Z coordinate of touch point
Q333=+0	;PRESET	Set the display in Z to 0
Q423=4	;NO. OF PROBE POINTS	Measure circle with 4 probes
Q365=0	;TYPE OF TRAVERSE	Move on circular path between measuring points
3 CALL PGM 35K47		Call part program
4 END PGM CYC413 MM		

Touch Probe Cycles: Automatic Presetting | Example: Presetting on top surface of workpiece and in center of a bolt hole circle

15

15.15 Example: Presetting on top surface of workpiece and in center of a bolt hole circle

The control is to write the measured bolt-hole circle center to the preset table so that it may be used at a later time.
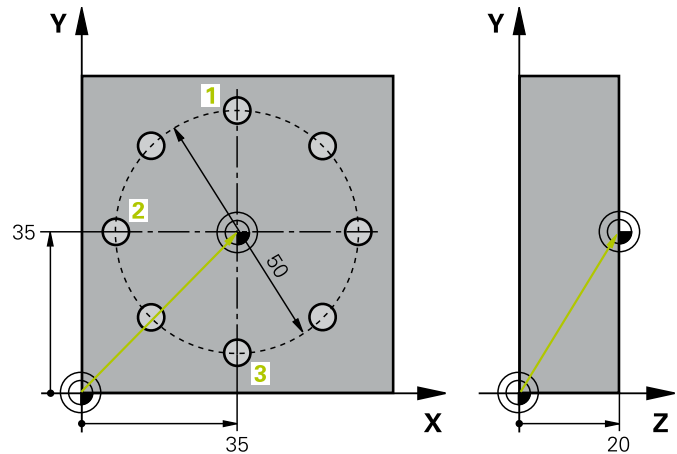
0 BEGIN PGM CYC416 MM		
1 TOOL CALL 69 Z		
2 TCH POBE 417 PRESET IN TS AXIS		Cycle definition for presetting in the touch probe axis
Q263=+7.5	;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+7.5	;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25	;1ST POINT 3RD AXIS	Touch point: Z coordinate
Q320=0	;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+50	;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1	;NUMBER IN TABLE	Write Z coordinate in line 1
Q333=+0	;PRESET	Set touch-probe axis to 0
Q303=+1	;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated preset referenced to the machine-based coordinate system (REF system)
3 TCH PROBE 416 PRESET CIRCLE CENTER		
Q273=+35	;CENTER IN 1ST AXIS	Center of the bolt hole circle: X coordinate
Q274=+35	;CENTER IN 2ND AXIS	Center of the bolt hole circle: Y coordinate
Q262=50	;NOMINAL DIAMETER	Diameter of the bolt hole circle
Q291=+90	;ANGLE OF 1ST HOLE	Polar coordinate angle for 1st hole center **1**
Q292=+180	;ANGLE OF 2ND HOLE	Polar coordinate angle for 2nd hole center **2**
Q293=+270	;ANGLE OF 3RD HOLE	Polar coordinate angle for 3rd hole center **3**
Q261=+15	;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+10	;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1	;NUMBER IN TABLE	Write center of bolt hole circle (X and Y) to row 1
Q331=+0	;PRESET	
Q332=+0	;PRESET	

Q303=+1	;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated preset referenced to the machine-based coordinate system (REF system)
Q381=0	;PROBE IN TS AXIS	Do not set a preset in the touch probe axis
Q382=+0	;1ST CO. FOR TS AXIS	No function
Q383=+0	;2ND CO. FOR TS AXIS	No function
Q384=+0	;3RD CO. FOR TS AXIS	No function
Q333=+0	;PRESET	No function
Q320=0	;SET-UP CLEARANCE.	Safety clearance in addition to SET_UP column
4 CYCL DEF 247 PRESETTING		Activate new preset with Cycle 247
Q339=1	;PRESET NUMBER	
6 CALL PGM 35KLZ		Call part program
7 END PGM CYC416 MM		

16

Touch Probe Cycles: Automatic Workpiece Inspection

16.1 Fundamentals

Overview

> ### *NOTICE*
>
> **Danger of collision!**
>
> When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
>
> ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
>
> ▶ Reset any coordinate transformations beforehand

> ⚙ The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.
>
> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

The control offers twelve cycles for measuring workpieces automatically.

Soft key	Cycle	Page
	0 REFERENCE PLANE Measuring a coordinate in a selectable axis	470
	1 POLAR PRESET Measuring a point in a probing direction	471
	420 MEASURE ANGLE Measuring an angle in the working plane	472
	421 MEASURE HOLE Measuring the position and diameter of a hole	475
	422 MEASURE CIRCLE OUTSIDE Measuring the position and diameter of a circular stud	479
	423 MEASURE RECTANGLE INSIDE Measuring the position, length, and width of a rectangular pocket	483
	424 MEASURE RECTANGLE OUTSIDE Measuring the position, length, and width of a rectangular stud	486

Soft key	Cycle	Page
425	425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width	489
426	426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge	492
427	427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis	495
430	430 MEASURE BOLT HOLE CIRCLE (2nd soft-key row) Measuring position and diameter of a bolt hole circle	498
431	431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane	501

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.

ℹ 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

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 (see figure at upper right). The highlighted result parameter belongs to that input parameter.
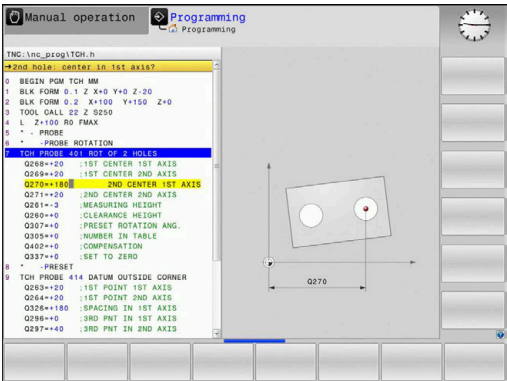
Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters Q180 to Q182.

Class of results	Parameter value
Measurement results are within tolerance	Q180 = 1
Rework is required	Q181 = 1
Scrap	Q182 = 1

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 because of 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

> **ℹ** This function works only:
> - If the tool table is active.
> - If tool monitoring is switched on in the cycle: Set **Q330** unequal to 0 or enter a tool name. To enter the tool name, press the associated soft key. The control no longer displays the right single quotation mark.
>
> If you perform several compensation measurements, the control adds the respective measured deviation to the value stored in the tool table.

Milling tool: If you refer to a milling cutter in parameter Q330, the appropriate values are compensated in the following way: The control basically always compensates the tool radius in column DR of the tool table, even if the measured deviation is within the specified tolerance. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: rework required).

Tool breakage monitoring

> **ℹ** This function works only:
> - If the tool table is active.
> - If tool monitoring is switched on in the cycle (set Q330 unequal to 0)
> - If the breakage tolerance RBREAK for the tool number entered in the table is greater than 0
>
>   **Further information:** User's Manual for Setup, Testing and Running NC Programs

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.

16.2 REFERENCE PLANE (Cycle 0, ISO: G55, software option 17)

Cycle run

1 In a 3-D 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.
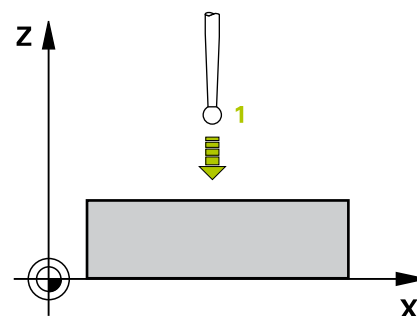
Please note while programming:

NOTICE

Danger of collision!

The control moves the touch probe in a 3-D 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 in order to avoid a collision when the programmed pre-positioning point is approached

---

Cycle parameters

▶ **Parameter number for result?**: Enter the number of the Q parameter to which you want to assign the coordinate. Input range: 0 to 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. Confirm with the **ENT** key. Input range: All NC axes

▶ **Position value?**: Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input range: -99999.9999 to 99999.9999

▶ To confirm your input, press the **ENT** key.

Example

67 TCH PROBE 0.0 REF. PLANE Q5 X-
68 TCH PROBE 0.1 X+5 Y+0 Z-5

16.3   POLAR PRESET (Cycle 1, software option 17)

Cycle run

Touch probe cycle 1 measures any position on the workpiece in any probing direction.

1   In a 3-D 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). Specify the probing direction by entering a polar angle 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.
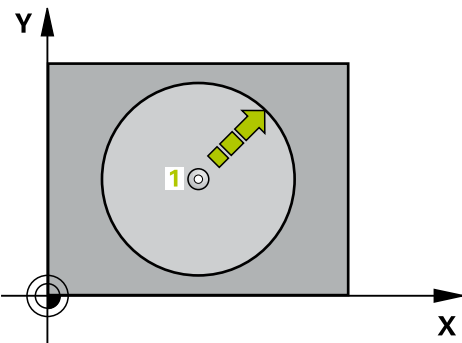
Please note while programming:

NOTICE
Danger of collision!
The control moves the touch probe in a 3-D 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 in order to avoid a collision when the programmed pre-positioning point is approached

> 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

▶   **Probing axis?**: Specify the probing axis with the axis key or the alphabetic keyboard. Confirm with the **ENT** key. Input range: **X**, **Y** or **Z**

▶   **Probing angle?**: Angle, measured from the probing axis, at which the touch probe is to move. Input range: -180.0000 to 180.0000

▶   **Position value?**: Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input range: -99999.9999 to 99999.9999

▶   To confirm your input, press the **ENT** key.

Example

67 TCH PROBE 1.0 POLAR PRESET
68 TCH PROBE 1.1 X ANGLE: +30
69 TCH PROBE 1.2 X+5 Y+0 Z-5

16.4 MEASURE ANGLE (Cycle 420, ISO: G420, software option 17)

Cycle run

Touch Probe Cycle 420 measures the angle that any straight surface on the workpiece describes with respect to the reference axis of the working plane.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), 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.

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:
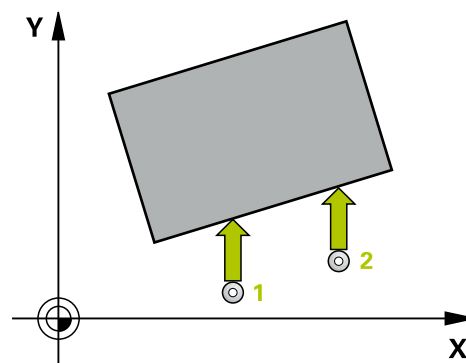
Parameter number	Meaning
Q150	The measured angle is referenced to the reference axis of the machining plane.

Please note while programming:

> **i** Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> 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**.

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis
3: Touch probe axis = measuring axis

▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
–1: Negative traverse direction
+1: Positive traverse direction

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** (incremental): 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. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
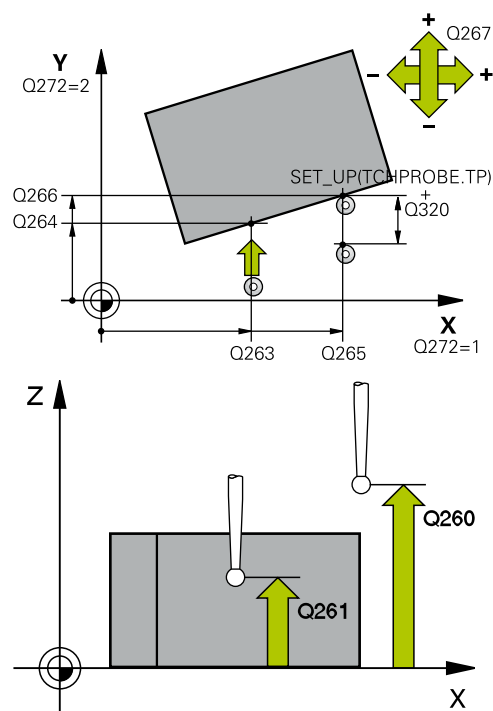
Example

5 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

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
0: Do not create a measuring log
1: Create a measuring log: The control saves the **log file named TCHPR420.TXT** in the same folder as 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**)

16.5 MEASURE HOLE (Cycle 421, ISO: G421, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the SET_UP column of the touch probe table.

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:

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

Please note while programming:

ⓘ   Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°

Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters

- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range: 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000
- ▶ **Q247 Intermediate stepping angle?** Q247 (incremental): 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°. Input range: -120.000 to 120.000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

Example

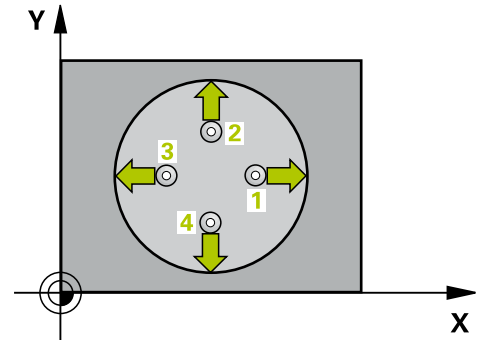| 5 TCH PROBE 421 MEASURE HOLE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q275 Maximum limit of size for hole?**: Maximum permissible diameter for the hole (circular pocket). Input range: 0 to 99999.9999

▶ **Q276 Minimum limit of size?**: Minimum permissible diameter for the hole (circular pocket). Input range: 0 to 99999.9999

▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Definition of whether the control should 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 the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

Q301=1	;MOVE TO CLEARANCE
Q275=75.12	;MAXIMUM LIMIT
Q276=74.95	;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

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

▶ **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
4: Use 4 measuring points (default setting)
3: Use 3 measuring points

▶ **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (Q301=1):
0: Move in a straight line between machining operations
1: Move in a circular arc on the pitch circle diameter between machining operations

▶ Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

16.6 MEAS. CIRCLE OUTSIDE (Cycle 422, ISO: G422, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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:

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

Please note while programming:

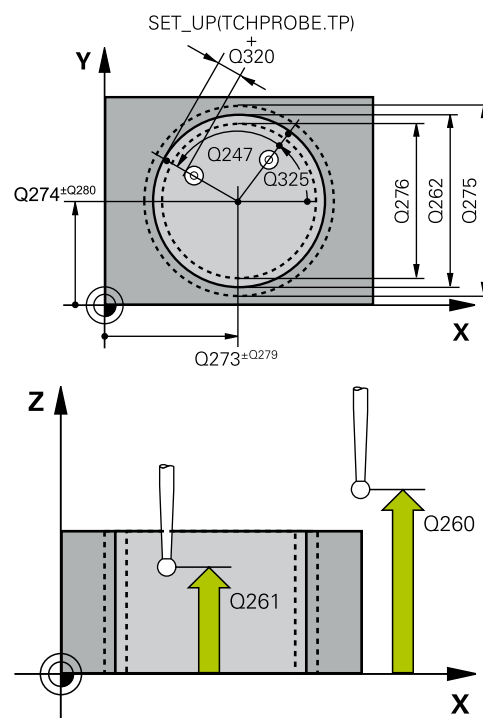**ℹ** Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The smaller the stepping angle, the less accurately the control can calculate the dimensions of the stud. Minimum input value: 5°.

Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters

▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q262 Nominal diameter?**: Enter the diameter of the stud. Input range: 0 to 99999.9999

▶ **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000

▶ **Q247 Intermediate stepping angle?** (incremental): 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°. Input range: -120.0000 to 120.0000

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

Example

5 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

▶ **Q277 Maximum limit of size for stud?**: Maximum permissible diameter for the stud. Input range: 0 to 99999.9999

▶ **Q278 Minimum limit of size for stud?**: Minimum permissible diameter for the stud. Input range: 0 to 99999.9999

▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
0: Do not create a measuring log
1: Create a measuring log: The control saves the **log file named TCHPR422.TXT** in the same folder as 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**.

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Tool number in the TOOL.T tool table

▶ **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
4: Use 4 measuring points (default setting)
3: Use 3 measuring points

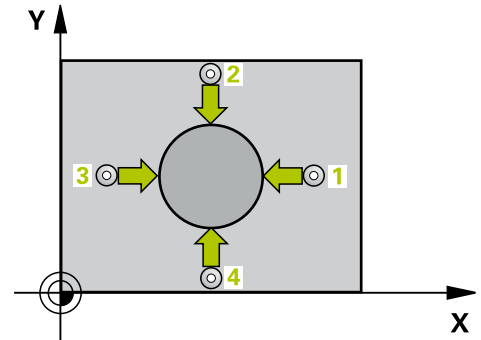|	
Q279=0.05	;TOLERANCE 1ST CENTER
Q280=0.05	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

▶ **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (Q301=1):
0: Move in a straight line between machining operations
1: Move in a circular arc on the pitch circle diameter between machining operations

▶ Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Touch Probe Cycles: Automatic Workpiece Inspection | MEASURE RECTANGLE INSIDE (Cycle 423, ISO: G423, software option 17)

16

16.7 MEASURE RECTANGLE INSIDE (Cycle 423, ISO: G423, software option 17)

Cycle run

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.

1   The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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:

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

Please note while programming:

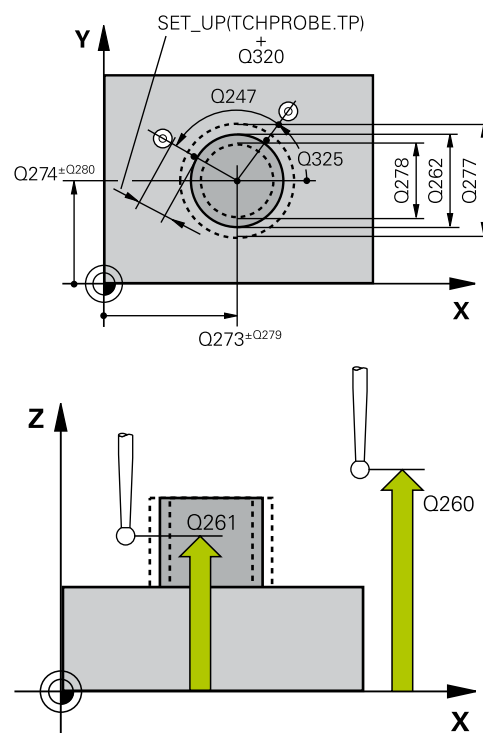🛈   Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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.

Cycle parameters

▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the pocket in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the pocket in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q282 1st side length (nominal value)?**: Pocket length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q283 2nd side length (nominal value)?**: Pocket length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q284 Max. size limit 1st side length?**: Maximum permissible length of the pocket. Input range: 0 to 99999.9999

▶ **Q285 Min. size limit 1st side length?**: Minimum permissible length of the pocket. Input range: 0 to 99999.9999

▶ **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the pocket. Input range: 0 to 99999.9999
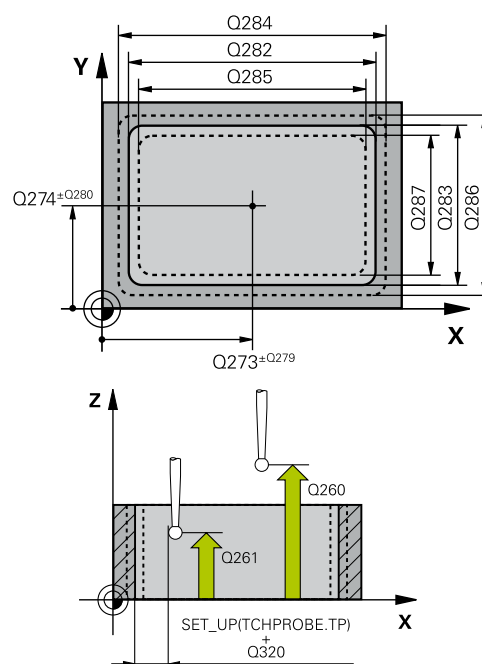
Example

5 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

▶ **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the pocket. Input range: 0 to 99999.9999

▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
0: Do not create a measuring log
1: Create a measuring log: The control saves the **log file named TCHPR423.TXT** in the same folder as 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**.

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Tool number in the TOOL.T tool table

Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

16.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, ISO: G424, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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:

| 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

Please note while programming:

ℹ Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

Touch Probe Cycles: Automatic Workpiece Inspection | MEASURE RECTANGLE OUTSIDE (Cycle 424, ISO: G424, software option 17)

16

Cycle parameters

- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- ▶ **Q282 1st side length (nominal value)?**: Stud length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999
- ▶ **Q283 2nd side length (nominal value)?**: Stud length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- ▶ **Q284 Max. size limit 1st side length?**: Maximum permissible length of the stud. Input range: 0 to 99999.9999
- ▶ **Q285 Min. size limit 1st side length?**: Minimum permissible length of the stud. Input range: 0 to 99999.9999

Example

5 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 Max. size limit 2nd side length?**: Maximum permissible width of the stud. Input range: 0 to 99999.9999

▶ **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the stud. Input range: 0 to 99999.9999

▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should 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** by default in the directory that also contains the associated .h file.
2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

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

Touch Probe Cycles: Automatic Workpiece Inspection | MEASURE INSIDE WIDTH (Cycle 425, ISO: G425, software option 17)

16

16.9 MEASURE INSIDE WIDTH (Cycle 425, ISO: G425, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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:

Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Please note while programming:

> ℹ️ Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
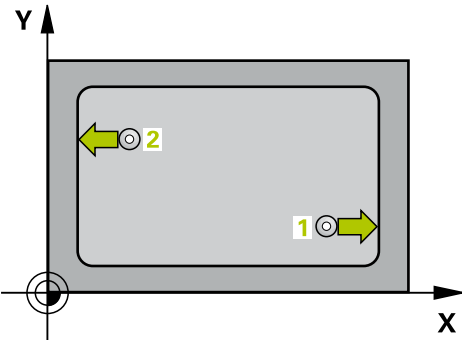
Cycle parameters
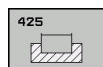
- ▶ **Q328 Starting point in 1st axis?** (absolute): Starting point for probing in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q329 Starting point in 2nd axis?** (absolute): Starting point for probing in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- ▶ **Q310 Offset for 2nd measuremnt (+/-)?** (incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the control does not offset the touch probe. Input range: -99999.9999 to 99999.9999

- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis

- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

- ▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

- ▶ **Q311 Nominal length?** : Nominal value of the length to be measured. Input range: 0 to 99999.9999

- ▶ **Q288 Maximum limit of size?**: Maximum permissible length. Input range: 0 to 99999.9999

- ▶ **Q289 Minimum limit of size?**: Minimum permissible length. Input range: 0 to 99999.9999

- ▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should 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** by default in the directory that also contains the associated .h file.
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
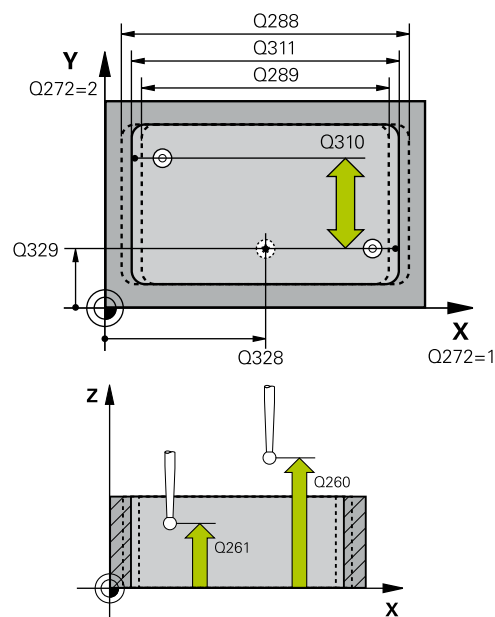
Example

5 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

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

▶ **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

16.10 MEASURE RIDGE WIDTH (Cycle 426, ISO: G426, software option 17)

**Cycle run**

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.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.

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:

| Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Please note while programming:

> **ℹ** Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q311 Nominal length?** : Nominal value of the length to be measured. Input range: 0 to 99999.9999

▶ **Q288 Maximum limit of size?**: Maximum permissible length. Input range: 0 to 99999.9999

▶ **Q289 Minimum limit of size?**: Minimum permissible length. Input range: 0 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
0: Do not create a measuring log
1: Create a measuring log: The control saves the **log file named TCHPR426.TXT** in the same folder as 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**.
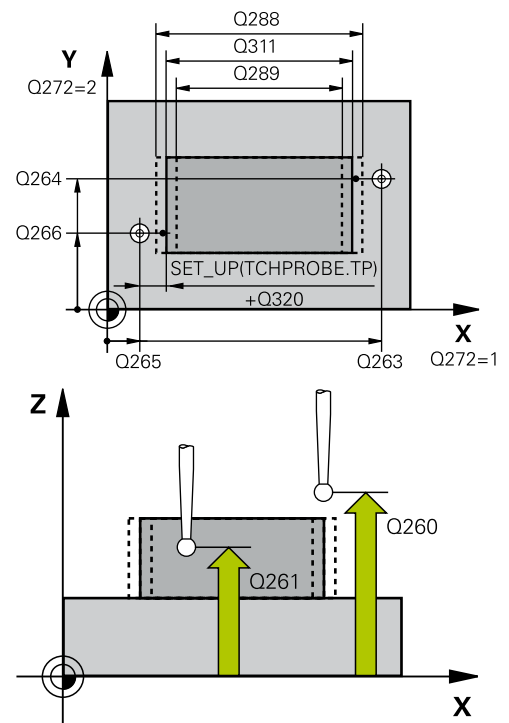
Example

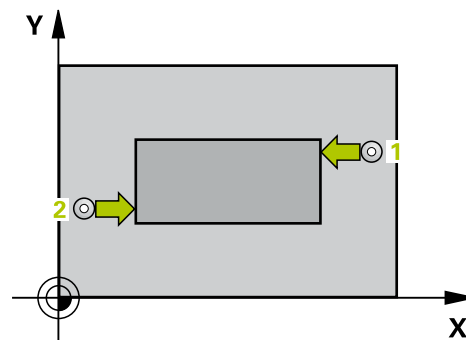| 5 TCH PROBE 426 MEASURE RIDGE WIDTH	
Q263=+50	;1ST POINT 1ST AXIS
Q264=+25	;1ST POINT 2ND AXIS
Q265=+50	;2ND PNT IN 1ST AXIS
Q266=+85	;2ND PNT IN 2ND AXIS
Q272=2	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q311=45	;NOMINAL LENGTH
Q288=45	;MAXIMUM LIMIT
Q289=44.95	;MINIMUM LIMIT
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

16.11 MEASURE COORDINATE (Cycle 427, ISO: G427, software option 17)

Cycle run

Touch probe cycle 427 determines a coordinate in a selectable axis and saves the value in a system 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.

1   The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction.

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:
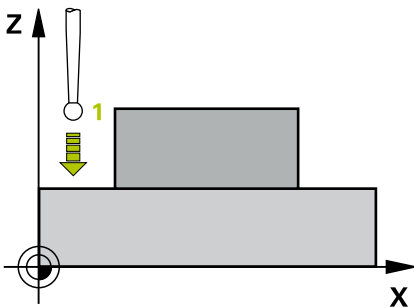
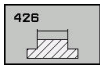| Parameter number	Meaning
Q160	Measured coordinate

Please note while programming:

🛈   Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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.

Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis
3: Touch probe axis = measuring axis

▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
–1: Negative traverse direction
+1: Positive traverse direction

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
0: Do not create a measuring log
1: Create a measuring log: The control saves the **log file named TCHPR427.TXT** in the same folder as the associated NC program.
2: Interrupt the program run and display the measuring log on the screen.Resume the NC program run with **NC Start**.

▶ **Q288 Maximum limit of size?**: Maximum permissible measured value. Input range: -99999.9999 to 99999.9999

▶ **Q289 Minimum limit of size?**: Minimum permissible measured value. Input range: -99999.9999 to 99999.9999
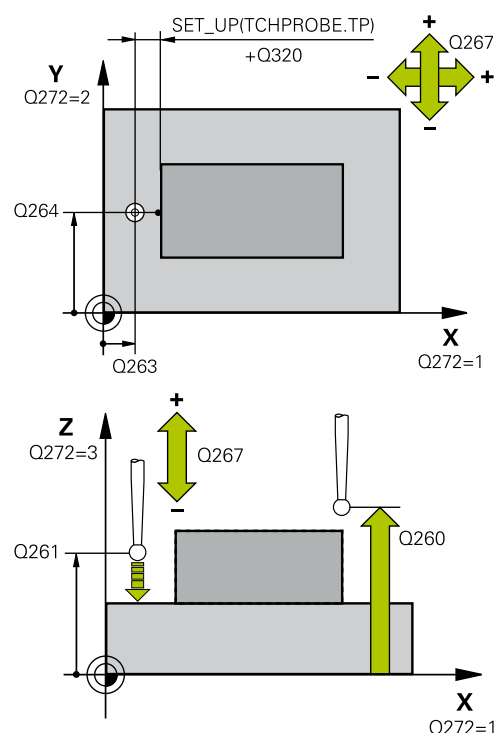
Example

| 5 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

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

▶ Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

16.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, ISO: G430, software option 17)

Cycle run

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.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to the programmed center point of the first hole **1**.

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:

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

Touch Probe Cycles: Automatic Workpiece Inspection | MEASURE BOLT HOLE CIRCLE (Cycle 430, ISO: G430, software option 17)

16

Please note while programming:

> ℹ Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> Cycle 430 only monitors for tool breakage; there is no automatic tool compensation.

Cycle parameters

▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range: 0 to 99999.9999

▶ **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range: -360.0000 to 360.0000

▶ **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range: -360.0000 to 360.0000

▶ **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range: -360.0000 to 360.0000

▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q288 Maximum limit of size?**: Maximum permissible diameter of bolt hole circle. Input range: 0 to 99999.9999

Example

5 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 Minimum limit of size?**: Minimum permissible diameter of bolt hole circle. Input range: 0 to 99999.9999

▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999

▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should 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 directory 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**.

▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
0: Do not interrupt program run, do not display an error message
1: Interrupt program run and display an error message

▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 468). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
0: Monitoring inactive
>0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.

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

16.13 MEASURE PLANE (Cycle 431, ISO: G431, software option 17)

Cycle run

Touch probe cycle 431 finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

1 The control positions the touch probe at rapid traverse (value from the **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 355), to 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.

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:

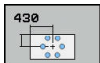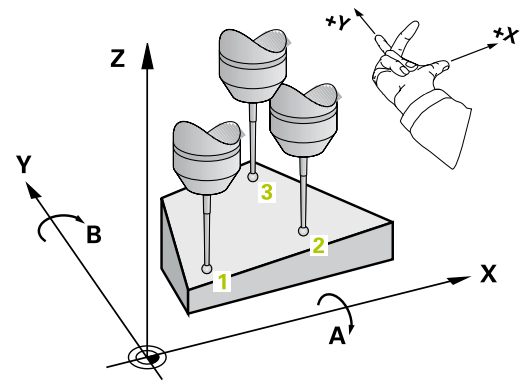| Parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to Q175	Measured values in the touch probe axis (first to third measurement)

Please note while programming:

ℹ Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control can only calculate the angular values if the three measuring points are not positioned on a straight line.

The spatial angles that are needed for tilting the working plane are saved in parameters Q170 – Q172. With the first two measuring points you also specify the direction of the reference 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.

NOTICE

Danger of collision!

When you enter the values into the preset table and then tilt the tool by programming the spatial angles SPA=0; SPB=0; SPC=0, there are multiple solutions in which the tilting axes are at 0.

▶ Make sure to program SYM (SEQ) + or SYM (SEQ) -

Cycle parameters

▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999

▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q295 2nd measuring point in 3rd axis?**
(absolute): Coordinate of the second touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999

▶ **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

▶ **Q298 3rd measuring point in 3rd axis?** (absolute): Coordinate of the third touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should 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 directory 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**.
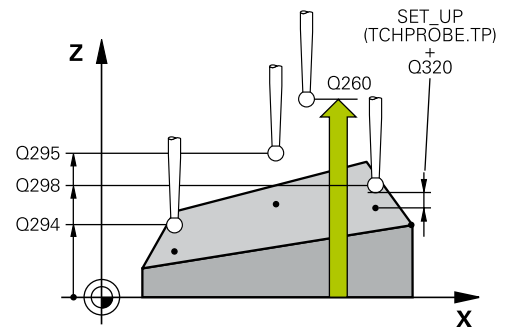
Example

5 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

16.14 Programming Examples

Example: Measuring and reworking a rectangular stud

Program run

- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account
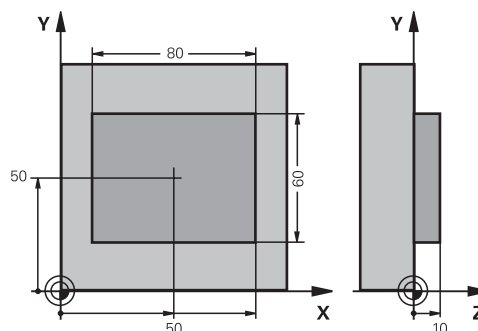
0 BEGIN PGM BEAMS MM	
1 TOOL CALL 69 Z	Tool call: roughing
2 L Z+100 R0 FMAX	Retract the tool
3 FN 0: Q1 = +81	Rectangle length in X (roughing dimension)
4 FN 0: Q2 = +61	Rectangle length in Y (roughing dimension)
5 CALL LBL 1	Call subprogram for machining
6 L Z+100 R0 FMAX	Retract the tool
7 TOOL CALL 99 Z	Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS.	Measure the rough-milled rectangle
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+50 ;CENTER IN 2ND AXIS	
Q282=80 ;FIRST SIDE LENGTH	Nominal length in X (final dimension)
Q283=60 ;2ND SIDE LENGTH	Nominal length in Y (final dimension)
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+30 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=0 ;MAX. LIMIT 1ST SIDE	Input values for tolerance checking not required
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	No measuring log transmission
Q309=0 ;PGM STOP TOLERANCE	Do not output an error message
Q330=0 ;TOOL	No tool monitoring
9 FN 2: Q1 = +Q1 - +Q164	Calculate length in X including the measured deviation
10 FN 2: Q2 = +Q2 - +Q165	Calculate length in Y including the measured deviation
11 L Z+100 R0 FMAX	Retract the touch probe

12 TOOL CALL 1 Z S5000	Tool call: finishing
13 CALL LBL 1	Call subprogram for machining
14 L Z+100 R0 FMAX M2	Retract the tool, end program
15 LBL 1	Subprogram with rectangular stud machining cycle
16 CYCL DEF 213 STUD FINISHING	
Q200=20 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE FOR MILLNG	
Q203=+10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=Q1 ;FIRST SIDE LENGTH	Length in X variable for roughing and finishing
Q219=Q2 ;2ND SIDE LENGTH	Length in Y variable for roughing and finishing
Q220=0 ;CORNER RADIUS	
Q221=0 ;ALLOWANCE IN 1ST AXS	
17 CYCL CALL M3	Cycle call
18 LBL 0	End of subprogram
19 END PGM BEAMS MM	

Example: Measuring a rectangular pocket and recording the results
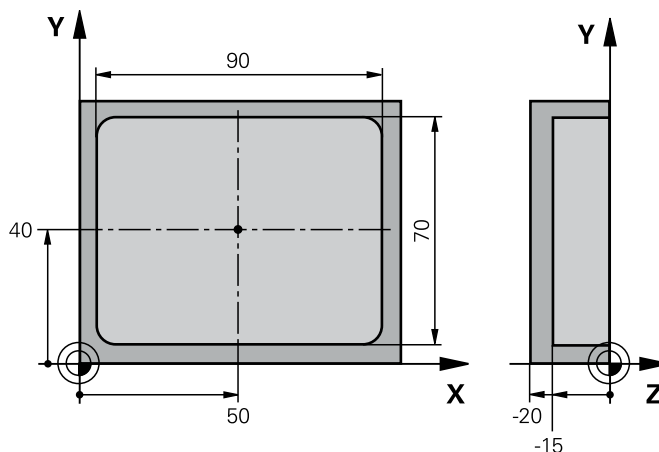
0 BEGIN PGM BSMEAS MM	
1 TOOL CALL 1 Z	Tool call: touch probe
2 L Z+100 R0 FMAX	Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50	;CENTER IN 1ST AXIS	
Q274=+40	;CENTER IN 2ND AXIS	
Q282=90	;FIRST SIDE LENGTH	Nominal length in X
Q283=70	;2ND SIDE LENGTH	Nominal length in Y
Q261=-5	;MEASURING HEIGHT	
Q320=0	;SET-UP CLEARANCE	
Q260=+20	;CLEARANCE HEIGHT	
Q301=0	;MOVE TO CLEARANCE	
Q284=90.15	;MAX. LIMIT 1ST SIDE	Maximum limit in X
Q285=89.95	;MIN. LIMIT 1ST SIDE	Minimum limit in X
Q286=70.1	;MAX. LIMIT 2ND SIDE	Maximum limit in Y
Q287=69.9	;MIN. LIMIT 2ND SIDE	Minimum limit in Y
Q279=0.15	;TOLERANCE 1ST CENTER	Permissible position deviation in X
Q280=0.1	;TOLERANCE 2ND CENTER	Permissible position deviation in Y
Q281=1	;MEASURING LOG	Save measuring log to a file
Q309=0	;PGM STOP TOLERANCE	Do not display an error message in case of a tolerance violation
Q330=0	;TOOL	No tool monitoring
4 L Z+100 R0 FMAX M2	Retract the tool, end program
5 END PGM BSMEAS MM	

17

Touch Probe Cycles: Special Functions

17.1 Fundamentals

Overview

> **NOTICE**
>
> **Danger of collision!**
>
> When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
>
> ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
>
> ▶ Reset any coordinate transformations beforehand

> ⚙ The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.
>
> HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.

The control provides cycles for the following special purposes:

Soft key	Cycle	Page
	3 **MEASURING** Measuring cycle for defining OEM cycles	509
	4 **MEASURING IN 3-D** Measuring any position	511
	441 **FAST PROBING** Measuring cycle for defining various touch probe parameters	526

17.2 MEASURE (Cycle 3, software option 17)

Cycle run

Touch probe cycle 3 measures any position on the workpiece in a selectable probing direction. Unlike other measuring 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.

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.

Please note while programming:

⚙ The exact behavior of Touch Probe Cycle 3 is defined by your machine tool builder or a software manufacturer who uses it within specific touch probe cycles.

ℹ The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other measuring 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 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.

With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle runs through the probe input X12 or X13.

Cycle parameters
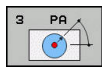
▶ **Parameter number for result?**: Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input range: 0 to 1999

▶ **Probing axis?**: Enter the axis in whose direction the probe is to move and confirm with the **ENT** key. Input range: X, Y, or Z

▶ **Probing angle?**: Angle, measured from the defined **probing axis** in which the touch probe is to move. Confirm with **ENT**. Input range: -180.0000 to 180.0000

▶ **Maximum measuring range?**: Enter the maximum traverse distance from the starting point by which the touch probe is to move. Confirm with ENT. Input range: -99999.9999 to 99999.9999

▶ **Feed rate measurement**: Enter the feed rate for measurement in mm/min. Input range: 0 to 3000.000

▶ **Maximum retraction distance?**: Traverse distance in the direction opposite 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 range: 0 to 99999.9999

▶ **Reference system? (0=ACT/1=REF)**: Define whether the probing direction and measuring result should reference the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**):
0: Probe in the current system and save the measuring result to the **ACT** system
1: Probe in the machine-based REF system and save the measuring result to the REF system.

▶ **Error mode? (0=OFF/1=ON)**: Specify whether the control is to 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 fourth result parameter and continues the cycle:
0: Display error message
1: Do not display error message

Example

4 TCH PROBE 3.0 MEASURING

5 TCH PROBE 3.1 Q1

6 TCH PROBE 3.2 X ANGLE: +15

7 TCH PROBE 3.3 ABST +10 F100 MB1 REFERENCE SYSTEM: 0

8 TCH PROBE 3.4 ERRORMODE1

17.3   MEASURING IN 3-D (Cycle 4, software option 17)

Cycle run

ℹ   Cycle 4 is an auxiliary cycle that can be used for probing with any touch probe (TS, TT or TL). The control does not provide a cycle for calibrating the TS touch probe in any probing direction.

Touch probe cycle 4 measures any position on the workpiece in the probing direction defined by a vector. Unlike other measuring 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 determining the measured value.

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.

Please note while programming:

ℹ   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.

Ensure during pre-positioning that the control moves the probe-tip center without compensation to the defined position!

Remember that the control always writes to four successive Q parameters.

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!

▶   Make sure that all touch points can be reached.

Cycle parameters
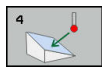
- ▶ **Parameter number for result?**: Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input range: 0 to 1999

- ▶ **Relative measuring path in X?**: X component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999

- ▶ **Relative measuring path in Y?**: Y component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999

- ▶ **Relative measuring path in Z?**: Z component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999

- ▶ **Maximum measuring range?**: Enter the maximum traverse distance from the starting point by which the touch probe may move along the direction vector. Input range: -99999.9999 to 99999.9999

- ▶ **Feed rate measurement**: Enter the feed rate for measurement in mm/min. Input range: 0 to 3000.000

- ▶ **Maximum retraction distance?**: Traverse distance in the direction opposite the probing direction, after the stylus was deflected. Input range: 0 to 99999.9999

- ▶ **Reference system? (0=ACT/1=REF)**: Specify whether the result of probing is to be saved in the input coordinate system (**ACT**), or with respect to the machine coordinate system (**REF**):
  0: Save measuring result to the **ACT** system
  1: Save measuring result to the **REF** system

Example

4 TCH PROBE 4.0 MEASURING IN 3-D
5 TCH PROBE 4.1 Q1
6 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
7 TCH PROBE 4.3 ABST+45 F100 MB50 REFERENCE SYSTEM:0

17.4 Calibrating a touch trigger probe

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.

---

i   Always calibrate a touch probe in the following cases:
- Commissioning
- 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 probe system directly after the calibration process.
The updated tool data is immediately effective. It is not necessary to repeat the tool call.

During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The control provides calibration cycles for calibrating the length and the radius:

▶ Press the **TOUCH PROBE** soft key

[CALIBRATE TS soft key icon]   ▶ Display the calibration cycles: Press the **CALIBRATE TS** soft key.
                     ▶ Select the calibration cycle

Calibration cycles provided by the control

Soft key	Function	Page
461	Calibrating the length	519
462	Measure the radius and the center offset using a calibration ring	521
463	Measuring the radius and the center offset using a stud or a calibration pin	523
460	Measure the radius and the center offset using a calibration sphere	515

17.5 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 values in columns **CAL_OF1** (reference axis) and **CAL_OF2** (minor axis) of the touch probe table. You can display the values on the screen by pressing the TOUCH PROBE TABLE soft key.

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. When running a touch probe cycle in the Manual Operation mode, the control saves the measuring log under the name TCHPRMAN.html. This file is stored in the folder TNC:\ *.
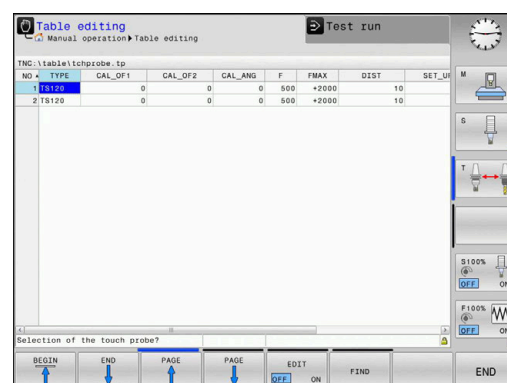
> **i** Ensure that the tool number of the tool table and the touch-probe number of the touch-probe table are correct. This is regardless of whether you want to use a touch-probe cycle in automatic mode or **Manual operation** mode.

> **📖** For more information, see Chapter Touch-probe table

17.6 CALIBRATE TS (Cycle 460, ISO: G460, software option 17)

Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration sphere. 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 sphere.

With Cycle 460 you can calibrate a triggering 3-D touch probe automatically using an exact calibration sphere.

It is also possible to capture 3-D calibration data. Software option 92, 3D-ToolComp, is required for this purpose. 3-D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3-D calibration data is stored under TNC:\system \CAL_TS<T_no.>_<T_idx.>.3DTC . The DR2TABLE column of the tool table references the 3DTC table. The 3-D calibration data is then taken into account when probing.

Cycle run

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 control first moves 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 radius calibration begins

7 Finally, the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.
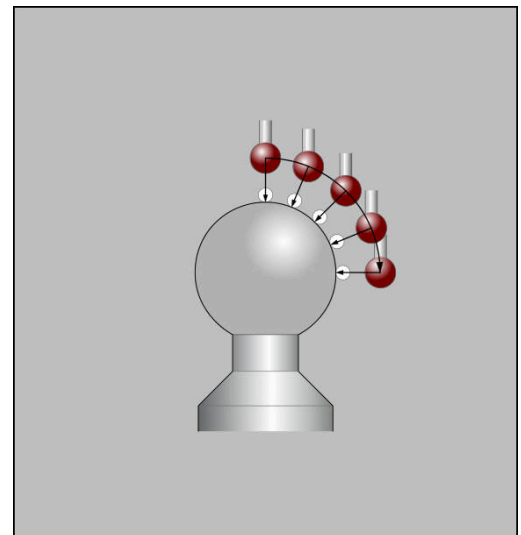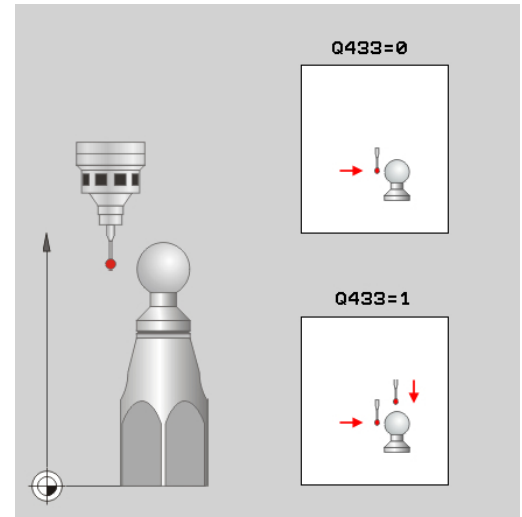
Radius and length calibration Q433=1

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 control first moves 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 radius calibration begins

7 The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

8 The control determines the length of the touch probe at the north pole of the calibration sphere.

9 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 3-D calibration.

3-D 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 thus determined

4 You can specify the number of probing 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 3-D 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.

Please note while programming:

> ### *NOTICE*
>
> **Danger of collision!**
>
> When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
>
> ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
>
> ▶ Reset any coordinate transformations beforehand

> ⚙ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

> ℹ 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 (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.
>
> Before a cycle definition you must program a tool call to define the touch-probe axis.
>
> Pre-position the touch probe so that it is located approximately above the center of the calibration sphere.
>
> If you program Q455=0, the control will not perform a 3-D calibration.
>
> If you program Q455=1- 30, the control will perform a 3-D 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 3-D calibration.
>
> If you program Q455=1- 30, a table will be stored under TNC:\Table\CAL_TS<T_no.>_<T_Idx.>.3DTC. <T_no> is the number of the touch probe, and <Idx> is its index.
>
> 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.

▶ **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999

▶ **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999

▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
0: Move to measuring height between measuring points
1: Move to clearance height between measuring points

▶ **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8

▶ **Q380 Ref. angle in ref. axis?** (absolute) Enter the Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range: 0 to 360.0000

▶ **Q433 Calibrate length (0/1)?**: Define whether the control is to calibrate the touch probe length after radius calibration, as well:
0: Do not calibrate touch probe length
1: Calibrate touch probe length

▶ **Q434 Preset for length?** (absolute): Coordinate of the calibration sphere center. This value must only be defined required if length calibration is to be carried out. Input range: -99999.9999 to 99999.9999

▶ **Q455 No. of points for 3-D calibrtn.?** Enter the number of touch points for 3-D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3-D calibration. During 3-D 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 3-D calibration. Input range: 1 to 30

Example

5 TCH PROBE 460 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.

17.7 CALIBRATE TS LENGTH (Cycle 461, ISO: G461, software option 17)

Cycle run

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.

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.
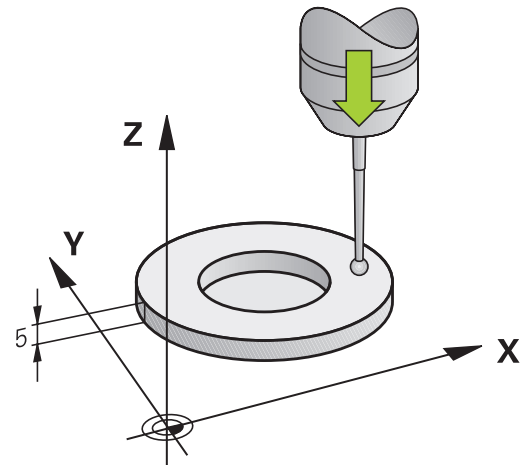
Please note while programming:

> ### *NOTICE*
>
> **Danger of collision!**
>
> When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
>
> ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
>
> ▶ Reset any coordinate transformations beforehand

> ⚙ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

> ⓘ The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.
>
> Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

▶ **Q434 Preset for length?** (absolute): Preset for the length (e.g. height of the calibration ring). Input range: -99999.9999 to 99999.9999
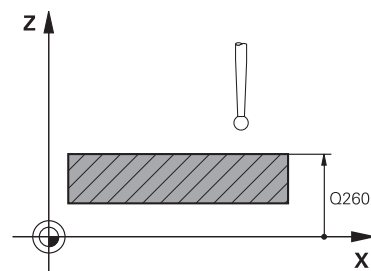
Example

5 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH
Q434=+5   ;PRESET

17.8 CALIBRATE TS RADIUS INSIDE (Cycle 462, ISO: G462, software option 17)

Cycle run

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 probing from opposite orientations, 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 operations. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.

- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Please note while programming:

> ### *NOTICE*
>
> **Danger of collision!**
>
> When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
>
> ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
>
> ▶ Reset any coordinate transformations beforehand

> **i** Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> 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.

> **⚙** In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer. Please refer to the machine manual.
>
> The orientation property of your touch probe is already predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.
>
> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

▶ **Q407 RING RADIUS** Enter the radius of the calibration ring. Input range 0 to 9.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8

▶ **Q380 Ref. angle in ref. axis?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: 0 to 360.0000
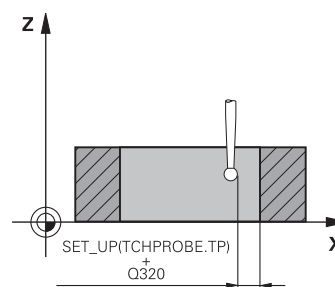
Example

5 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

17.9 CALIBRATE TS RADIUS OUTSIDE (Cycle 463, ISO: G463, software option 17)

Cycle run

Before starting the calibration cycle, you need to preposition 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 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 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 operations. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.

- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Please note while programming:

> ### *NOTICE*
>
> **Danger of collision!**
>
> When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
>
> ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
>
> ▶ Reset any coordinate transformations beforehand

> **ℹ** Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> 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.

> **⚙** In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer. Please refer to the machine manual.
>
> The orientation property of your touch probe is already predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.
>
> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

- ▶ **Q407 Radius of calibr. stud?**: Diameter of the calibration ring or pin. Input range: 0 to 99.9999
- ▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- ▶ **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- ▶ **Q380 Ref. angle in ref. axis?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: 0 to 360.0000

Example

5 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

17.10 FAST PROBING (Cycle 441, ISO G441, software option 17)

Cycle run

You can use the touch probe cycle 441 to globally specify various touch probe parameters (e.g. the positioning feed rate) for all subsequently used touch probe cycles.

Please note while programming:

> Cycle 441 sets parameters for touch probe cycles. This cycle does not perform machine movements
>
> **END PGM**, **M2**, **M30** resets the global settings of Cycle 441.
>
> Cycle parameter **Q399** depends on your machine configuration. Orienting the touch probe system via the NC program must be enabled by your machine tool builder.
>
> The feed rate can also be limited by your machine tool builder. The absolute maximum feed rate is defined in the machine parameter **maxTouchFeed** (No. 122602).
>
> Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate, also with Q397=1, only with the potentiometer for feed motions.

Cycle parameters

▶ **Q396 Positioning feed rate?**: Define the feed rate at which the touch probe will be moved to the specified positions. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**

▶ **Q397 Pre-pos. at machine's rapid?**: Define whether the control, when pre-positioning the touch probe, traverses at **FMAX** feed rate (machine's rapid traverse):
0: Pre-position with feed rate from **Q396**
1: Pre-position with machine's rapid traverse **FMAX**Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate, also with Q397=1, only with the potentiometer for feed motions. The feed rate can also be limited by your machine tool builder. The absolute maximum feed rate is defined in the machine parameter **maxTouchFeed** (No. 122602).

▶ **Q399 Angle tracking (0/1)?**: Define whether the control orients the touch probe before each probing operation:
0: No orientation
1: Orient spindle before each probing operation (improves the accuracy)

▶ **Q400 Automatic interruption?** Define whether the control interrupts the program run and outputs the measuring results on the screen following a measuring cycle for automatic workpiece measurement:
0: Do not interrupt program run even if, in the specific probing cycle, the output of measuring results on the screen is selected
1: Interrupt program run and output measuring results on the screen. You can then resume the NC program run with **NC Start**.

Example

5 TCH PROBE 441 FAST PROBING	
Q 396=3000	;POSITIONING FEED RATE
Q 397=0	;SELECT FEED RATE
Q 399=1	;ANGLE TRACKING
Q 400=1	;INTERRUPTION

18

Touch Probe
Cycles: Automatic
Kinematics
Measurement

18.1 Kinematics measurement with TS touch probes (KinematicsOpt option)

Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts need to be manufactured with precision and reproducible accuracy even over extended periods.

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 at right) and the kinematic conditions actually existing on the machine (see **2** in the figure at right). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure at right). 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 to really meet these complex requirements: a 3-D 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.
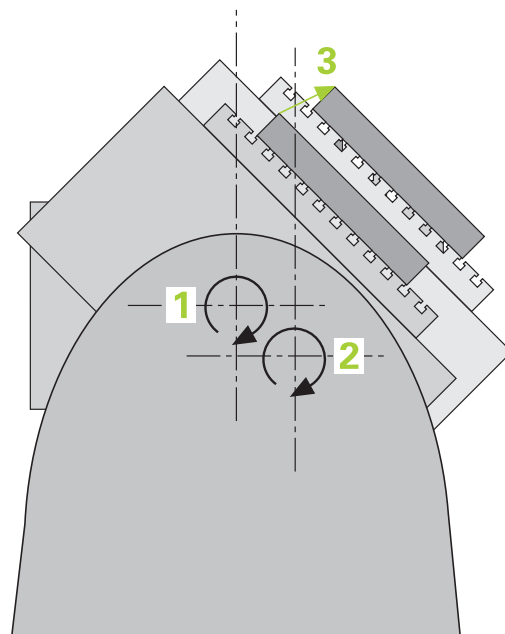
Overview

The control offers the following cycles that enable you to automatically save, restore, check, and optimize the machine kinematics:

Soft key	Cycle	Page
	450 **SAVE KINEMATICS** Automatic saving and restoring of kinematic configurations	534
	451 **MEASURE KINEMATICS** Automatic checking or optimizing of machine kinematics	537
	452 **PRESET COMPENSATION** Automatic checking or optimizing of machine kinematics	552

18.2 Prerequisites

The following are prerequisites for using the KinematicsOpt option:

- The software options 48 (KinematicsOpt), 8 (Software option 1) and 17 (Touch Probe function) must be enabled.
- The 3-D 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. 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 kinematics description of the machine must be complete and correct. The transformation values must be entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine tool builder during commissioning).
- The machine tool builder 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.

Please note while programming:

NOTICE
Danger of collision!
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.
▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
▶ Reset any coordinate transformations beforehand

⚙ HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

🛈 If an M function has been defined in machine parameter **mStrobeRotAxPos** (no. 204803), 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.

18.3 SAVE KINEMATICS (Cycle 450, ISO: G450, option)

Cycle run

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.
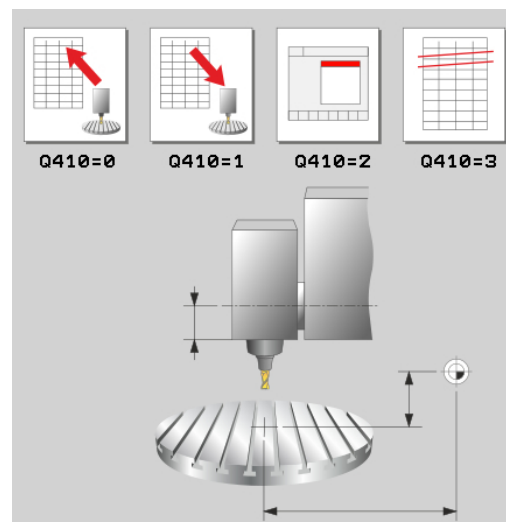
Please note while programming:

> **i** Always save the active kinematics configuration 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. Define the preset again, if necessary.
>
> The cycle will not restore identical values, but only data that is different from the existing data. Compensations can only be restored if they had been saved before.

> **⚙** Only save and restore data with Cycle 450, while no tool carrier kinematics configuration that includes transformations is active.

Cycle parameters
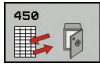
▶ **Q410 Mode (0/1/2/3)?**: Define whether you wish to backup or restore the kinematics configuration:
0: Backup active kinematics configuration
1: Restore saved kinematics configuration
2: Display current memory status
3: Delete a data record.

▶ **Q409/QS409 Name of data record?**: Number or name of the record designator. When specifying a number, you can enter values from 0 to 99999, and when entering letters the string length may not exceed 16 characters. 16 memory locations in total are available. Q409 has no 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 data blocks after confirmation (mode 3). You can use the following wildcards for searching:
?: Any single character
$: Any single alphabetic character (letter)
#: Any single number
*: Any character string of any length

Saving the current kinematics

5 TCH PROBE 450 SAVE KINEMATICS	
Q410=0	;MODE
Q409=947	;MEMORY DESIGNATION

Restoring data blocks

5 TCH PROBE 450 SAVE KINEMATICS	
Q410=1	;MODE
Q409=948	;MEMORY DESIGNATION

Displaying all saved data blocks

5 TCH PROBE 450 SAVE KINEMATICS	
Q410=2	;MODE
Q409=949	;MEMORY DESIGNATION

Deleting data blocks

5 TCH PROBE 450 SAVE KINEMATICS	
Q410=3	;MODE
Q409=950	;MEMORY DESIGNATION

Logging function

After running Cycle 450, the control creates a log (**tchpr450.txt**) 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.

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 is removed, too. If the data in the file is changed manually, the data records may become corrupted so that they are unusable.

> **ⓘ** If the **TNC:\table\DATA450.KD** file does not exist, it is generated automatically when Cycle 450 is executed.
>
> Make sure that you delete empty files with the name **TNC:\table\DATA450.KD**, if any, before starting Cycle 450. If there is an empty storage table (**TNC:\table\DATA450.KD**) that does not contain any lines yet, an error message will be displayed upon starting to run Cycle 450. In this case, delete the empty storage 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).

18.4 MEASURE KINEMATICS (Cycle 451, ISO: G451, option)

Cycle run

Touch probe cycle 451 enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

> **ℹ** 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 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.

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 the Program Run operating mode and start the calibration program.
4　The control automatically measures all rotary axes successively in the resolution you defined.
5　The control saves the measured values in the following Q parameters:

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).

▶ So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration sphere

▶ Also ensure that there is enough space to reach the safety 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 tool builder, enabling the control to move the rotary axes. The machine tool builder must have entered the number of the M function in machine parameter **mStrobeRotAxPos** (no. 244803) for this purpose.

▶ Note the documentation of the machine tool builder

Define a retraction height greater than 0 if software 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°

Defining the number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1- 2), e.g. 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 machine with rotary tables/tilting tables: Clamp the calibrating ball 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.

Notes on the accuracy

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 of results recorded by the control in the measuring 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.

> **i** If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.
>
> 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 tool manual provides further information.

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 near the rotary axis angle at which the workpiece is to be machined.
  - 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.

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 made 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.

ⓘ 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 determine the rotary axis backlash (see "Logging function", Page 551).

Backlash measurement is not possible if an M function for positioning the rotary axes is set in machine parameter **mStrobeRotAxPos** (no. 204803) or if the axis is a Hirth axis.

Please note while programming:

> **ℹ** 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.
>
> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.
>
> 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.
>
> If machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to -1 (M function positions the rotary axes), only start a measurement if all rotary axes are at 0°.
>
> 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.
>
> For angle optimization, the machine manufacturer must change the configuration correspondingly.
>
> 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.
>
> Compensation of the angle is only possible with option #52 **KinematicsComp**.

ⓘ If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification**), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

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 machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Programming in inches: The control always records the log data and results of measurement in millimeters.

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.

Cycle parameters

▶ **Q406 Mode (0/1/2/3)?**: Specify whether the control should check or optimize the active kinematics:
0: Check active machine kinematics. The control measures the kinematics in the rotary axes you have defined, but it does not make any changes to it. The control displays the results of measurement in a measurement log.
1: Optimize active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **the position of the rotary axes** of the active kinematics.
2: Optimize active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **angle and positioning errors.** Software option 52, KinematicsComp, is required for the compensation of angle errors.
3: Optimize active machine kinematics: The control automatically corrects the machine datum. It then optimizes **angle and positioning errors.** Software option 52, KinematicsComp, is required.

▶ **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999 Alternatively **PREDEF**

▶ **Q408 Retraction height?** (absolute): Input range: 0.0001 to 99999.9999
0: Do not move to retraction height. The control moves the touch probe 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.

▶ **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**

▶ **Q380 Ref. angle in ref. axis?** (absolute) Enter the Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range: 0 to 360.0000

Saving and checking the kinematics

4 TOOL CALL "TCH PROBE" Z	
5 TCH PROBE 450 SAVE KINEMATICS	
Q410=0	;MODE
Q409=5	;MEMORY DESIGNATION
6 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	;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
Q431=0	;PRESET
Q432=0	;BACKLASH, ANG. RANGE

- ▶ **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range: -359.999 to 359.999

- ▶ **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made. Input range: -359.999 to 359.999

- ▶ **Q413 Angle of incidence in A axis?**: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999

- ▶ **Q414 No. of meas. points in A (0...12)?**: Number of probe measurements to be used for measuring the A axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12

- ▶ **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range: -359.999 to 359.999

- ▶ **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made. Input range: -359.999 to 359.999

- ▶ **Q417 Angle of incidence in B axis?**: Inclination angle in the B axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999

- ▶ **Q418 No. of meas. points in B (0...12)?**: Number of probe measurements to be used for measuring the B axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12

- ▶ **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range: -359.999 to 359.999

- ▶ **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made. Input range: -359.999 to 359.999

- ▶ **Q421 Angle of incidence in C axis?**: Inclination angle in the C axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999

▶ **Q422 No. of meas. points in C (0...12)?**: Number of probe measurements to be used for measuring the C axis. Input range: 0 to 12. If the input value = 0, the control does not measure the respective axis.

▶ **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Input range: 3 to 8. Less measuring points increase speed and more measuring points increase measurement precision.

▶ **Q431 Preset (0/1/2/3)?** Define whether the control automatically sets the active preset to the center of the sphere:
0: Do not define the preset automatically at the center of the sphere: Preset manually before cycle start
1: Define the preset automatically at the center of the sphere (this overwrites the active preset): Manually preposition the touch probe before cycle start using the calibration sphere
2: Automatically define the preset at the center of the sphere after measuring): Preset manually before cycle start
3: Define the preset before and after measurement at the center of the sphere (the active preset will be overwritten): Preposition the touch probe manually before cycle start using the calibration sphere

▶ **Q432 Angular range of backlash comp.?**: Here, you define the angle value to be used as traversing angle when measuring 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 range: -3.0000 to +3.0000

ℹ️ 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.

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 is 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.

Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

1 TOOL CALL "TCH PROBE" Z	
2 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=0	;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

Logging function

After running Cycle 451, the control will create a log **(TCHPR451.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
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - 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)

## 18.5 PRESET COMPENSATION (Cycle 452, ISO: G452, option)

Cycle run

Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, ISO: G451, option)", Page 537). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the calibration sphere after optimization.

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 use Cycle 451 to finally 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.
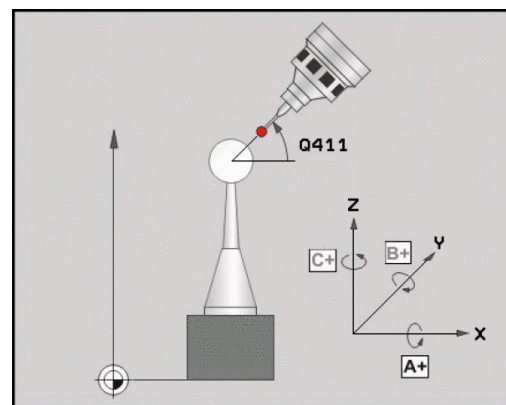
Parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–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

Please note while programming:

> **ⓘ** 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.
>
> In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.
>
> Note that all functions for tilting in the working plane are reset.
>
> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.
>
> Before defining the cycle, you must set the preset in 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 probe monitoring is inactive.
>
> 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.

> **ⓘ** If the kinematics data determined exceed the permissible limit (**maxModification**), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.
>
> Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.
>
> 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 machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.
>
> Programming in inches: The control always records the log data and results of measurement in millimeters.

Cycle parameters

▶ **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999

▶ **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

▶ **Q408 Retraction height?** (absolute): Input range: 0.0001 to 99999.9999
0: Do not move to retraction height. The control moves the touch probe 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.

▶ **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**

▶ **Q380 Ref. angle in ref. axis?** (absolute) Enter the Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range: 0 to 360.0000

▶ **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range: -359.999 to 359.999

▶ **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made. Input range: -359.999 to 359.999

▶ **Q413 Angle of incidence in A axis?**: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999

▶ **Q414 No. of meas. points in A (0...12)?**: Number of probe measurements to be used for measuring the A axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12

Calibration program

4 TOOL CALL "TCH PROBE" Z	
5 TCH PROBE 450 SAVE KINEMATICS	
Q410=0	;MODE
Q409=5	;MEMORY DESIGNATION
6 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

▶ **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range: -359.999 to 359.999

▶ **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made. Input range: -359.999 to 359.999

▶ **Q417 Angle of incidence in B axis?**: Inclination angle in the B axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999

▶ **Q418 No. of meas. points in B (0...12)?**: Number of probe measurements to be used for measuring the B axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12

▶ **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range: -359.999 to 359.999

▶ **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made. Input range: -359.999 to 359.999

▶ **Q421 Angle of incidence in C axis?**: Inclination angle in the C axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999

▶ **Q422 No. of meas. points in C (0...12)?**: Number of probe measurements to be used for measuring the C axis. Input range: 0 to 12. If the input value = 0, the control does not measure the respective axis.

▶ **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Input range: 3 to 8. Less measuring points increase speed and more measuring points increase measurement precision.

▶ **Q432 Angular range of backlash comp.?**: Here, you define the angle value to be used as traversing angle when measuring 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 range: -3.0000 to +3.0000

Adjustment of interchangeable heads

The goal of this procedure is to achieve that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, a fork head is adjusted to the A and C axes. 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.

▶ Set preset (using Q431 = 2 or 3 in Cycle 451) after measuring the reference head.

Measuring a reference head

1 TOOL CALL "TCH PROBE" Z	
2 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

- ▶ 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

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

Adjusting an interchangeable head

3 TOOL CALL "TCH PROBE" Z
4 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

Drift compensation

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

▶ Set 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

1 TOOL CALL "TCH PROBE" Z	
2 CYCL DEF 247 PRESETTING	
Q339=1	;PRESET NUMBER
3 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.

> **ⓘ** This procedure can also be performed on machines without rotary axes.

Drift compensation

4 TOOL CALL "TCH PROBE" Z
5 TCH PROBE 452 PRESET COMPENSATION
Q407=12.5 ;SPHERE RADIUS
Q320=0 ;SET-UP CLEARANCE
Q408=0 ;RETR. HEIGHT
Q253=99999;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

Logging function

After running Cycle 452, the control creates a log (**TCHPR452.html**) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes
  - 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)

Notes on log data

(see "Logging function", Page 551)

19

Touch Probe
Cycles: Automatic
Tool Measurement

19.1 Fundamentals

Overview

> **ℹ** Operating notes
> - When running touch probe cycles, Cycle **8 MIRROR IMAGE**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active.
> - HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.

> **⚙** The control and the machine tool must be set up by the machine tool builder for use of the TT touch probe.
>
> Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.
>
> The touch probe cycles are available only with the Touch Probe Functions software option (option 17).

In conjunction with the control's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central TOOL.T tool file and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Measurement of a stationary tool
- Measurement of a rotating tool
- Measurement of individual teeth

You can program the cycles for tool measurement in **Programming** mode of operation using the **TOUCH PROBE** key. The following cycles are available:

New format	Old format	Cycle	Page
480 U U CAL.	30 U U CAL.	Calibrating the TT, Cycles 30 and 480	570
484 U U CAL.		Calibrating the wireless TT 449, Cycle 484	572
481	31	Measuring the tool length, Cycles 31 and 481	574
482	32	Measuring the tool radius, Cycles 32 and 482	576
483 U U	33 U U	Measuring the tool length and radius, Cycles 33 and 483	578

> **i** The measuring cycles can be used only when the central tool file TOOL.T is active.
>
> Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.

Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only two differences between Cycles 31 to 33 and Cycles 481 to 483:

- Cycles 481 to 483 are also available in controls for ISO programming under G481 to G483.
- Instead of a selectable parameter for the status of the measurement, the new cycles use the fixed parameter **Q199**.

Setting machine parameters

> ℹ️ Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200).
>
> Touch probe cycles 480, 481, 482, 483 and 484 can be hidden with the machine parameter **hideMeasureTT** (No. 128901).
>
> When measuring a stationary tool, the Control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius [mm]

The probing feed rate is calculated as follows:
v = measuring tolerance • n with

v:	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate:

probingFeedCalc (No. 122710) = **ConstantTolerance**:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance (**measureTolerance1** (no. 122715), the sooner you will encounter this effect.

probingFeedCalc (No. 122710) = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc (No. 122710) = **ConstantFeed**:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance = r • **measureTolerance1**/ 5 mm, where

r:	Active tool radius [mm]
measureTolerance1:	Maximum permissible error of measurement

Entries in the tool table TOOL.T

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT.	Cutting direction of the tool for measuring a rotating tool	**Cutting direction (M3 = -)?**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to **offsetToolAxis**. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill	4 (4 teeth)	R (offset required because tool diameter is larger than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from **offsetToolAxis** (no. 122707) used)
Radius cutter e.g. with a diameter of 10 mm	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured	5 (always define an offset at least equal to the tool radius in order to make sure that the measured diameter is correct)

19.2 Calibrating the TT (Cycle 30 or 480, ISO: G480 option 17)

Cycle run

The TT is calibrated with measuring cycle TCH PROBE 30 or TCH PROBE 480. (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 565). The calibration process runs automatically. The control also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylindrical pin. The resulting calibration values are stored in the control memory and are accounted for during subsequent tool measurement.

Calibration process:

1 Clamp the calibrating tool. The calibrating tool must be a precisely cylindrical part, for example a cylinder pin

2 Manually position the calibrating tool in the working plane via the center of the TT

3 Position the calibrating tool in the tool axis approx. 15 mm + safety clearance above the TT

4 The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.

5 The calibration process along the tool axis starts

6 Calibration then follows in the working plane

7 The control positions the calibrating tool in the working plane at a position of TT radius + set-up clearance + 11 mm

8 Then the TNC moves the tool downwards along the tool axis and the calibration process starts

9 During probing, the control moves in a square pattern

10 The control saves the calibration values and considers them during subsequent tool measurement

11 The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Please note while programming:

ℹ The functioning of the calibration cycle is dependent on machine parameter **CfgTTRoundStylus** (No. 114200). Refer to your machine manual.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the TOOL.T tool table.

The position of the TT within the machine working space must be defined by setting machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**.

If you change the setting of any of the machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**, you must recalibrate.

Cycle parameters
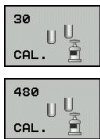
▶ **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistToolAx** (no. 114203)). Input range: -99999.9999 to 99999.9999

Example of old format

6 TOOL CALL 1 Z
7 TCH PROBE 30.0 CALIBRATE TT
8 TCH PROBE 30.1 HEIGHT: +90

Example of new format

6 TOOL CALL 1 Z
7 TCH PROBE 480 CALIBRATE TT
Q260=+100 ;CLEARANCE HEIGHT

19.3 Calibrating the wireless TT 449 (Cycle 484, ISO: G484, option 17)

Fundamentals

With Cycle 484, you can calibrate your tool touch probe, e.g the wireless infrared TT 449 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle 484 you must move the tool over the TT

Cycle run

To calibrate the tool touch probe, program measuring cycle TCH PROBE 484. In input parameter Q536, you can specify whether you want to run the cycle semi-automatically or fully automatically.

Semi-automatic—stop before running

- ▶ Insert the calibrating tool
- ▶ Define and start the calibration cycle
- ▶ The control interrupts the calibration cycle.
- ▶ The control displays a dialog in a new window.
- ▶ You are prompted to manually position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact.

Fully automatic—no stop before running

- ▶ Insert the calibrating tool
- ▶ Position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact.
- ▶ Define and start the calibration cycle
- ▶ The calibration cycle is executed without stopping. The calibration process starts from the current position of the tool.

Calibrating tool:

The calibrating tool must be a precisely cylindrical part, for example a cylindrical pin. Enter the exact length and radius of the calibrating tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Please note while programming:

> ### *NOTICE*
>
> **Danger of collision!**
>
> To avoid collisions the tool must be pre-positioned before calling the cycle with **Q536**=1! The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.
>
> ▶ Specify whether to stop before cycle start or run the cycle automatically without stopping.

> **i** The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.
>
> The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. When using a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. The use of a calibrating tool of too small a diameter and/or protruding too far from the chuck may cause significant inaccuracies.
>
> Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.
>
> The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

▶ **Q536 Stop before running (0=Stop)?**: Specify whether to stop before cycle start or run the cycle automatically without stopping:
0: Stop before running the cycle. You are prompted in a dialog to manually position the tool above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC Start to continue the calibration process or press the **CANCEL** soft key to cancel the calibration process
1: No stop before running the cycle. The control starts the calibration process from the current position. Before running Cycle 484, you must position the tool above the tool touch probe.

Example

6 TOOL CALL 1 Z
7 TCH PROBE 484 CALIBRATE TT
Q536=+0   ;STOP BEFORE RUNNING

19.4 Measuring tool length (Cycle 31 or 481, ISO: G481, option 17)

Cycle run

To measure the tool length, program the measuring cycle TCH PROBE 31 or TCH PROBE 481 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483"). Via input parameters you can measure the length of a tool by three methods:

■ If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.

■ If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is stationary.

■ If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is stationary.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**R-OFFS**).

Cycle for measuring a stationary tool (e.g. for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

Cycle for measuring individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). You can enter an additional offset in Tool offset: Length (**L-OFFS**) in the tool table. The control probes the tool radially while it is rotating to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function in Cycle TCH PROBE 31, set parameter Probe the teeth = 1.

Please note while programming:

> ℹ️ Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
>
> You can run an individual tooth measurement of tools with **up to 20 teeth**.

Cycle parameters

▶ **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
0: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL=0. If there is already a value in TOOL.T, it will be overwritten.
1: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T)
2: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q115. Nothing is entered under L or DL in the tool table.

▶ **Parameter number for result?**: Parameter number in which the control saves the status of measurement results:
0.0: Tool is within 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 in your NC program, answer the dialog prompt with **NO ENT**.

▶ **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: -99999.9999 to 99999.9999

▶ **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).

Measuring a rotating tool for the first time; old format

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1 q5
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1

Example of new format

6 TOOL CALL 12 Z	
7 TCH PROBE 481 CAL. TOOL LENGTH	
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH

19.5 Measuring a tool radius (Cycle 32 or 482, ISO: G482, option 17)

Cycle run

To measure a tool radius, program the measuring cycle TCH PROBE 32 or TCH PROBE 482 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 565). Select via input parameters by which of two methods the tool radius is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. The control probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Please note while programming:

> **i** Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
>
> The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.
>
> Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, define the number of teeth **CUT** as 0 in the tool table and adjust machine **CfgTT** (no. 122700). Refer to your machine manual.

Cycle parameters
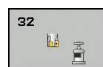
▶ **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
0: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR=0. If there is already a value in TOOL.T, it will be overwritten.
1: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T)
2: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q116. Nothing is entered under R or DR in the tool table.

▶ **Parameter number for result?**: Parameter number in which the control saves the status of the measurement results:
0.0: Tool is within 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 in your NC program, answer the dialog prompt with **NO ENT**.

▶ **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: -99999.9999 to 99999.9999

▶ **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).

Measuring a rotating tool for the first time; old format

6 TOOL CALL 12 Z
7 TCH PROBE 32.0 CAL. TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 0

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

6 TOOL CALL 12 Z
7 TCH PROBE 32.0 CAL. TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 1

Example of new format

6 TOOL CALL 12 Z
7 TCH PROBE 482 CAL. TOOL RADIUS
Q340=1   ;CHECK
Q260=+100 ;CLEARANCE HEIGHT
Q341=1   ;PROBING THE TEETH

19

Touch Probe Cycles: Automatic Tool Measurement | Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17)

19.6 Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17)

Cycle run

To measure both the length and radius of a tool, program the measuring cycle TCH PROBE 33 or TCH PROBE 483 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 565). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Via input parameters you can select the desired type of measurement:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for Cycles 31 and 32 as well as 481 and 482.

Please note while programming:

> **ℹ** Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
>
> The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.
>
> Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgTT** (No. 122700) Refer to your machine manual.

Touch Probe Cycles: Automatic Tool Measurement | Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17)

19

Cycle parameters

▶ **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
0: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL=0 and DR=0. If there is already a value in TOOL.T, it will be overwritten.
1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in Q parameters Q115 and Q116. If the delta value is greater than the permissible tool length or radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T)
2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The control calculates the deviation from the stored values and writes it to Q parameter Q115 or Q116. Nothing is entered under L, R, DL, or DR in the tool table.

▶ **Parameter number for result?**: Parameter number in which the control saves the status of the measurement results:
0.0: Tool is within tolerance
1.0: Tool is worn (**LTOL** and/or **RTOL** exceeded)
2,0: Tool is broken (**LBREAK** and/or **RBREAK** exceeded). If you do not wish to use the result of measurement in your NC program, answer the dialog prompt with **NO ENT**.

▶ **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: -99999.9999 to 99999.9999

▶ **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).

Measuring a rotating tool for the first time; old format

6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 0
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE TEETH: 0

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 1 q5
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE TEETH: 1

Example of new format

6 TOOL CALL 12 Z	
7 TCH PROBE 483 MEASURE TOOL	
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH

20

Tables of Cycles

20.1 Overview

Fixed cycles

Cycle number	Cycle name	DEF active	CALL active	Page
7	Datum shift	■		299
8	Mirroring	■		306
9	Dwell time	■		325
10	Rotation	■		308
11	Scaling factor	■		310
12	Program call	■		326
13	Oriented spindle stop	■		327
14	Contour definition	■		223
18	Thread cutting		■	345
19	Tilting the working plane	■		313
20	Contour data SL II	■		228
21	Pilot drilling SL II		■	230
22	Rough out SL II		■	232
23	Floor finishing SL II		■	236
24	Side finishing SL II		■	238
25	Contour train		■	241
26	Axis-specific scaling	■		311
27	Cylinder surface		■	267
28	Cylindrical surface slot		■	270
29	Cylinder surface ridge		■	274
32	Tolerance	■		328
39	Cylinder surface contour		■	277
200	Drilling		■	75
201	Reaming		■	77
202	Boring		■	79
203	Universal drilling		■	82
204	Back boring		■	88
205	Universal pecking		■	92
206	Tapping with a floating tap holder, new		■	117
207	Rigid tapping, new		■	120
208	Bore milling		■	100
209	Tapping with chip breaking		■	125
220	Polar pattern	■		211
221	Cartesian pattern	■		214

Cycle number	Cycle name	DEF active	CALL active	Page
225	Engraving		■	332
232	Face milling		■	338
233	Face milling (selectable milling direction, consider the side walls)		■	196
239	Ascertain the load	■		343
240	Centering		■	73
241	Single-lip deep-hole drilling		■	103
247	Presetting	■		305
251	Rectangular pocket (complete machining)		■	157
252	Circular pocket (complete machining)		■	163
253	Slot milling		■	170
254	Circular slot		■	175
256	Rectangular stud (complete machining)		■	181
257	Circular stud (complete machining)		■	186
258	Polygon stud		■	190
262	Thread milling		■	132
263	Thread milling/countersinking		■	136
264	Thread drilling/milling		■	140
265	Helical thread drilling/milling		■	144
267	Outside thread milling		■	148
270	Contour train data		■	250
275	Trochoidal slot		■	252
276	3-D contour train		■	245

Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	Reference plane	■		470
1	Polar preset	■		471
3	Measuring	■		509
4	Measuring in 3-D	■		511
30	Calibrate the TT	■		570
31	Measure/Inspect the tool length	■		574
32	Measure/Inspect the tool radius	■		576
33	Measure/Inspect the tool length and the tool radius	■		578
400	Basic rotation using two points	■		381
401	Basic rotation over two holes	■		384
402	Basic rotation over two studs	■		388
403	Compensate misalignment with rotary axis	■		393
404	Set basic rotation	■		398
405	Compensate misalignment with the C axis	■		399
408	Preset at slot center (FCL 3 function)	■		409
409	Preset at ridge center (FCL 3 function)	■		413
410	Preset from inside of rectangle	■		417
411	Preset from outside of rectangle	■		421
412	Preset from inside of circle (hole)	■		425
413	Preset from outside of circle (stud)	■		430
414	Preset from outside of corner	■		435
415	Preset from inside of corner	■		440
416	Preset from circle center	■		445
417	Preset in touch probe axis	■		450
418	Preset at center between four holes	■		452
419	Preset in any one axis	■		457
420	Workpiece—measure angle	■		472
421	Workpiece—measure hole (center and diameter of hole)	■		475
422	Workpiece—measure circle from outside (diameter of circular stud)	■		479
423	Workpiece—measure rectangle from inside	■		483
424	Workpiece—measure rectangle from outside	■		486
425	Workpiece—measure inside width (slot)	■		489
426	Workpiece—measure outside width (ridge)	■		492
427	Workpiece—measure in any selectable axis	■		495
430	Workpiece—measure bolt hole circle	■		498
431	Workpiece—measure plane	■		498

Cycle number	Cycle name	DEF active	CALL active	Page
441	Fast probing	■		526
450	KinematicsOpt: Save kinematics (option)	■		534
451	KinematicsOpt: Measure kinematics (option)	■		537
452	KinematicsOpt: Preset compensation	■		530
460	Calibrate the touch probe	■		515
461	Calibrate touch probe length	■		519
462	Calibrate touch probe inside radius	■		521
463	Calibrate touch probe outside radius	■		523
480	Calibrate the TT	■		570
481	Measure/Inspect the tool length	■		574
482	Measure/Inspect the tool radius	■		576
483	Measure/Inspect the tool length and the tool radius	■		578
484	Calibrate TT	■		572
1410	Probing on edge	■		370
1411	Probing two circles	■		374
1420	Probing in plane	■		366

Index

HEIDENHAIN

www.klartext-portal.com
The Information Site for
HEIDENHAIN Controls

Klartext App

The Klartext on Your
Mobile Device

Google
Play Store

Apple
App Store

Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the
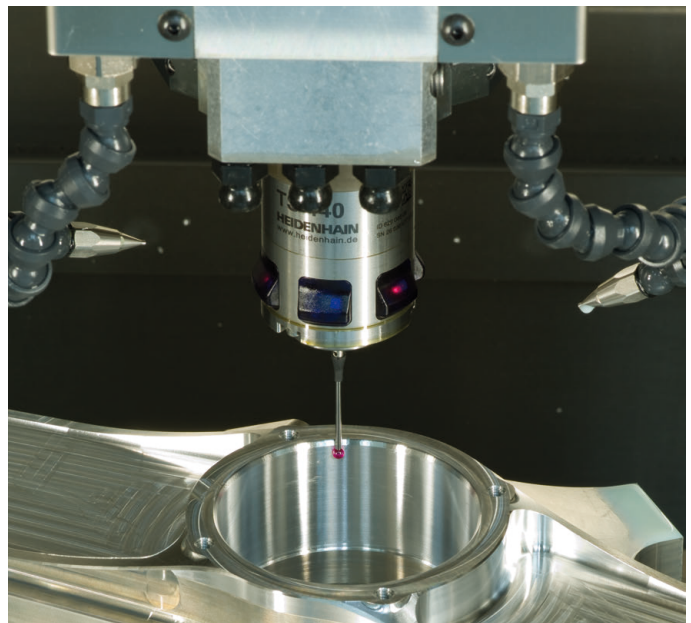dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220	Signal transmission by cable
TS 440, TS 444	Infrared transmission
TS 640, TS 740	Infrared transmission

- Workpiece alignment
- Setting presets
- Workpiece measurement

Tool touch probes

TT 140	Signal transmission by cable
TT 449	Infrared transmission
TL	Non-contacting laser systems

- Tool measurement
- wear monitoring
- tool breakage detection