HEIDENHAIN

TNC 620

User's Manual for Cycle Programming

NC Software
817600-07
817601-07
817605-07

English (en)
10/2019

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:

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

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

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

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

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

tnc-userdoc@heidenhain.de

1.2 Control model, software, and features

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

Control model	NC software number
TNC 620	817600-07
TNC 620 E	817601-07
TNC 620 Programming station	817605-07

The suffix E indicates the export version of the control. The following software options are unavailable or only available to a limited extent in the export version:

- Advanced Function Set 2 (option 9) limited to four-axis interpolation
- KinematicsComp (option 52)

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 HEIDENHAIN 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 ■ Set the preset in the **Manual operation** mode of operation ■ 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
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
Opt. contour milling (Option 167)	
Optimized contour cycles	■ Cycle 271: **OCM CONTOUR DATA** ■ Cycle 272: **OCM ROUGHING** ■ Cycle 273: **OCM FINISHING FLOOR** ■ Cycle 274: **OCM FINISHING SIDE**

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. For an overview of the optional Q parameters that have been added in this software version, refer to "New or changed cycle functions of software 81760x-07". 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.

Proceed as follows:

- Call the cycle definition
- Press the right cursor key until the new Q parameters are displayed
- Confirm the displayed default value
- Alternatively, enter the desired value
- To load 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 load 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.

1.3 New or changed cycle functions of software 81760x-06

- New Cycle 1410 PROBING ON EDGE (option 17), see "PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410, option 17)", Page 397
- New Cycle 1411 PROBING TWO CIRCLES (option 17),see "PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411, option 17)", Page 401
- New Cycle 1420 PROBING IN PLANE (option 17), see "PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420, option 17)", Page 393
- In Cycle 24 SIDE FINISHING, a tangential helix will be used for rounding in the last infeed step, see "SIDE FINISHING (Cycle 24, DIN/ISO: G124, option 19)", Page 267
- Cycle 233 FACE MILLING, was extended by parameter Q367, SURFACE POSITION, see "FACE MILLING (Cycle 233, DIN/ISO: G233, option 19)", Page 196
- Cycle 257 CIRCULAR STUD, now uses Q207 FEED RATE MILLING for roughing, too, see "CIRCULAR STUD (Cycle 257, DIN/ISO: G257, option 19)", Page 186
- The touch probe cycles 408 to 419 consider chkTiltingAxes (no. 204600) for presetting, see "Touch Probe Cycles: Automatic Presetting", Page 431
- 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 431
- 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, DIN/ISO: G420, option 17)", Page 498
- Cycle 450 SAVE KINEMATICS, no longer writes identical values when restoring, see "SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48)", Page 562
- In Cycle 451 MEASURE KINEMATICS, the value 3 was added to cycle parameter Q406 MODE, see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)", Page 565
- In Cycles 451 MEASURE KINEMATICS , the radius of the calibration sphere is monitored in the second measurement only, see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)", Page 565
- A REACTION column was added to the touch probe table, see "Touch-probe table", Page 380
- The CfgThreadSpindle machine parameter (no. 113600) is now available, see "TAPPING with a floating tap holder (Cycle 206, ISO: G206)", Page 119 , see "TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)", Page 122, see "TAPPING WITH CHIP BREAKING (Cycle 209, DIN/ISO: G209, option 19)", Page 126 , see "THREAD CUTTING (Cycle 18, DIN/ISO: G86, option 19)", Page 369

1.4 New or changed cycle functions of software 81760x-07

- New point pattern cycle 224 DATAMATRIX CODE PATTERN for the creation of a DataMatrix code, see "DATAMATRIX CODE PATTERN (Cycle 224, DIN/ISO: G224, option 19) ", Page 241
- New cycle 238 MEASURE MACHINE STATUS for monitoring machine components for wear, see "MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155)", Page 365
- New cycle 271 OCM CONTOUR DATA for defining machining information for the OCM cycles, see "OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167) ", Page 297
- New cycle 272 OCM ROUGHING for machining open pockets while maintaining the tool angle, see "OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167) ", Page 299
- New cycle 273 OCM FINISHING FLOOR for machining open pockets while maintaining the tool angle,see "OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167)", Page 302
- New cycle 274 OCM FINISHING SIDE for machining open pockets while maintaining the tool angle, see "OCM FINISHING SIDE (Cycle 274, DIN/ISO: G274, option 167)", Page 304

- New DATUM TABLE soft key in the Program Run, Single Block and Program Run, Full Sequence operating modes. In addition, transfer of actual values to the datum table is possible in the Program Run, Single Block and Program Run, Full Sequence operating modes. see "Editing the datum table in the Program Run, Single Block or Program Run, Full Sequence operating mode", Page 214
- In cycles 205 UNIVERSAL PECKING and 241 SINGLE-LIP D.H.DRLNG, the input value for Q379 STARTING POINT is checked and compared to Q201 DEPTH. In case of conflict, an error message will be issued, see "UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205, option 19)", Page 92 or Page 103
- Using cycle 225 ENGRAVING, it is possible to engrave the path or name of an NC program, see "Engraving the name and path of an NC program", Page 359
- If a limit has been programmed in cycle 233, the FACE MILLING cycle will extend the contour in the infeed direction by the corner radius, see "FACE MILLING (Cycle 233, DIN/ISO: G233, option 19)", Page 196
- Cycle 239 ASCERTAIN THE LOAD is only displayed if this has been defined by the machine tool builder, see "ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, option 143)", Page 367
- The help graphics for Q224 ANGLE OF ROTATION in cycle 256 RECTANGULAR STUD was changed, see "RECTANGULAR STUD (Cycle 256, DIN/ISO: G256, option 19)", Page 181
- The help graphics for Q326 SPACING IN 1ST AXIS and Q327 SPACING IN 2ND AXIS in cycle 415 PRESET INSIDE CORNER was changed, see "PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415, option 17)", Page 458
- The help graphics for Q341 PROBING THE TEETH in cycles 481 and 31 CAL. TOOL LENGTH and in cycles 482 and 32 CAL. TOOL RADIUS was changed, see "Measuring tool length (Cycle 31 or 481, DIN/ISO: G481, option 17)", Page 600 or Page 604
- In cycles 14xx, it is possible to use a handwheel for pre-positioning in semi-automatic mode. After probing, you can move to clearance height manually, see "Semi-automatic mode", Page 387

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

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

- Press the **CYCL DEF** key

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring and counterboring	74
DRILLING/ THREAD	Cycles for tapping, thread cutting and thread milling	118
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	208
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	250
PATTERN	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	234
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, determining the load,	346

▷

- 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

- Press the **TOUCH PROBE** key.

Soft key	Cycle group	Page
ROTATION	Cycles for automatic measurement and compensation of workpiece misalignment	383
PRESET	Cycles for automatic workpiece presetting	432
MEASURING	Cycles for automatic workpiece inspection	490
SPECIAL CYCLES	Special cycles	536
CALIBRATE TS	Touch probe calibration	543
KINEMATICS	Cycles for automatic kinematics measurement	559
TT CYCLES	Cycles for automatic tool measurement (enabled by the machine manufacturer)	592

▷

- Switch to machine-specific touch probe cycles, if available; these touch probe cycles can be integrated by the machine tool builder

3

Using Fixed Cycles

3.1 Working with fixed cycles

Machine-specific cycles (option 19)

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

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

do the following:

Proceed as follows:

- 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.

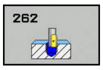
Further information: "Calling a cycle", Page 54

Defining a cycle using soft keys

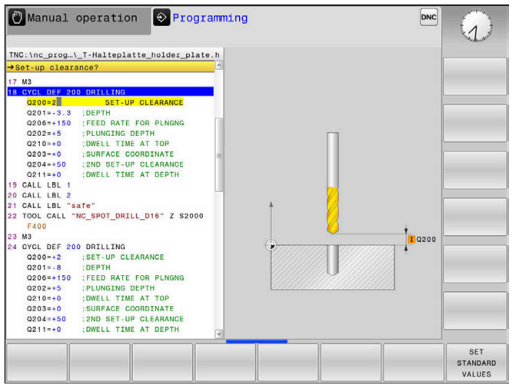
Proceed as follows:

- ▶ Press the **CYCL DEF** key
- \> The soft-key row shows the available groups of cycles.

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



- ▶ 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 the required parameters
- ▶ 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

Proceed as follows:

- ▶ Press the **CYCL DEF** key
- \> The soft-key row shows the available groups of cycles.

- ▶ Press the **GOTO** key
- \> The control displays a pop-up window with an overview of the cycles.
- ▶ Select the desired cycle with the cursor keys
- ▶ Alternatively enter the cycle number
- ▶ Confirm each input with the **ENT** key
- \> The control then initiates the cycle dialog as described above.

Example

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

Calling a cycle

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Spindle direction of rotation (miscellaneous function **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.

Proceed as follows:

CYCL CALL

- Press the **CYCL CALL** key
- Press the **CYCL CALL M** soft key
- If required, enter an M function (e.g. **M3**, to switch on the spindle)
- Press **END** to end the dialog

Calling a cycle with CYCL CALL PAT

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

Further information: "Pattern definition with PATTERN DEF", Page 61

Further information: "Point tables", Page 68

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

> 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**, proceed as follows:

- ▶ Program **M99** in the positioning block
- > The control moves to the last starting point.
- ▶ Alternatively, define a new machining cycle with **CYCL DEF**

> The control does not support **M89** in combination with free programming of contours!

Calling a cycle with SEL CYCLE

With **SEL CYCLE**, you can call any NC program as a machining cycle.

Proceed as follows:

PGM CALL

- Press the **PGM CALL** key
- Press the **SEL CYCLE** soft key
- Press the **SELECT FILE** soft key
- Select an NC program

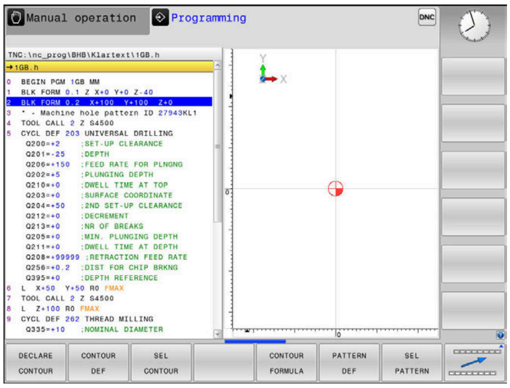
CYCL CALL

- Press the **CYCL CALL M**, **CYCL CALL PAT**, or **CYCL CALL POS** soft key
- Alternatively program **M99**

> When you execute an NC program selected with **SEL CYCLE**, it will be executed in the Program Run, Single Block operating mode without stopping after each NC block. In addition, it is visible as a single NC block in the Program Run, Full Sequence operating mode.
>
> Please note that **CYCL CALL PAT** and **CYCL CALL POS** use a positioning logic before executing the cycle. With respect to the positioning logic, **SEL CYCLE** and Cycle 12 **PGM CALL** show the same behavior: In point pattern cycles, the clearance height is calculated based on the maximum value of all Z positions existing at the start point of the pattern and all Z positions in the point pattern. With **CYCL CALL POS**, there will be no pre-positioning in the tool axis direction. This means that you need to manually program any pre-positioning in the file you call.

3.2 Program defaults for cycles

Overview

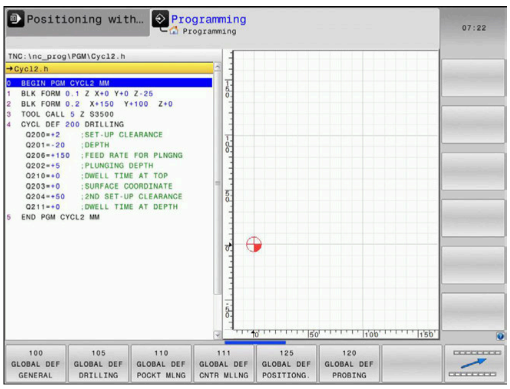


All Cycles 20 to 25, as well as all of those with numbers 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** you can define these cycle parameters at the beginning of the program, so that 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	59
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	59
110 GLOBAL DEF POCKT MLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	59
111 GLOBAL DEF CNTR MLLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	59
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	60
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	60

Entering GLOBAL DEF



Proceed as follows:

- Press the **PROGRAMMING** key

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

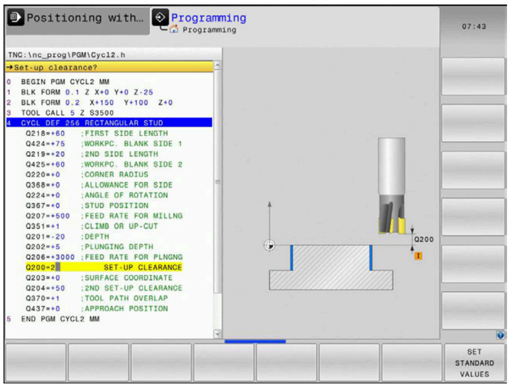
PROGRAM DEFAULTS
- Press the **PROGRAM DEFAULTS** soft key

GLOBAL DEF
- Press the **GLOBAL DEF** soft key

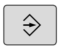
100 GLOBAL DEF GENERAL
- Select the desired GLOBAL DEF function, e.g. by pressing the **GLOBAL DEF GENERAL** soft key
- Enter the required definitions
- Press the **ENT** key each time to confirm

Using GLOBAL DEF information

If you entered the 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:



- Press the **PROGRAMMING** key

- Press the **CYCL 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** 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 moves the tool at the end of a machining step (the next machining position will be 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

- **RETRACT CHIP BREAKNG**: 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**: Climb or up-cut machining
- **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**: Climb or up-cut machining

> 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

- **SAFETY 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** option is activated
- **MOVE TO CLEARANCE**: 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.

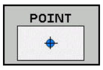
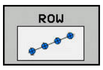
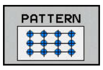
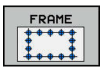
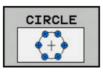
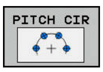
NOTICE

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!

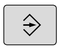
- Use **PATTERN DEF** only in connection with the tool axis **Z**

The following machining patterns are available:

Soft key	Machining pattern	Page
	POINT Definition of up to any 9 machining positions	63
	ROW Definition of a single row, straight or rotated	63
	PATTERN Definition of a single pattern, straight, rotated or distorted	64
	FRAME Definition of a single frame, straight, rotated or distorted	65
	CIRCLE Definition of a full circle	66
	PITCH CIRCLE Definition of a pitch circle	67

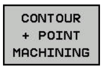
Entering PATTERN DEF

Proceed as follows:

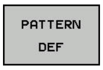


- Press the **PROGRAMMING** key

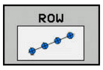
- Press the **SPEC FCT** key



- Press the **CONTOUR + POINT MACHINING** soft key



- Press the **PATTERN DEF** soft key



- Select the desired machining pattern, e.g. press the "single row" soft key
- Enter the required definitions
- Press the **ENT** key each time to confirm

Using PATTERN DEF

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

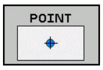
Further information: "Calling a cycle", Page 54

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

> **i**
>
> 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 set-up clearance and the 2nd set-up clearance reference the coordinate surface in PATTERN DEF.
>
> Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q352**=1. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.

Defining individual machining positions

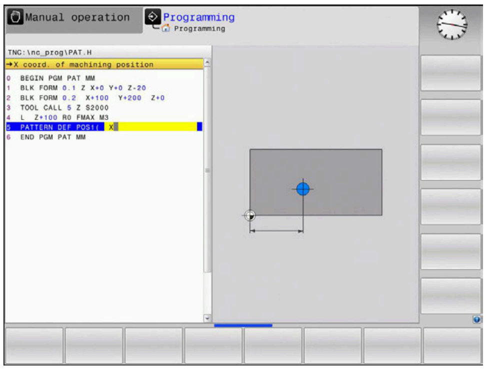
> You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.
>
> POS1 must be programmed with absolute coordinates. POS2 to POS9 can be programmed as absolute and/or incremental values.
>
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- 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

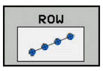
Example

```
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

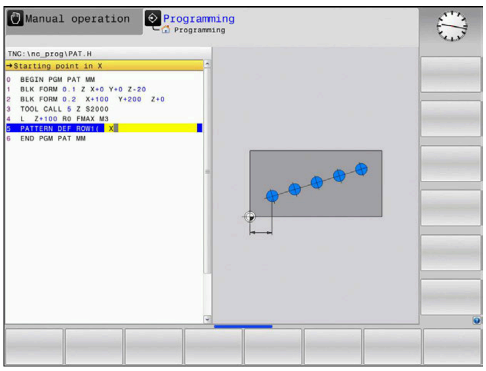
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **Starting point in X** (absolute): Coordinate of the 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

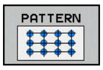
```
10 L  Z+100 R0  FMAX
11 PATTERN DEF  ROW1
    (X+25 Y+33.5  D+8 NUM5  ROT+0 Z
    +0)
```



Defining an individual 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.

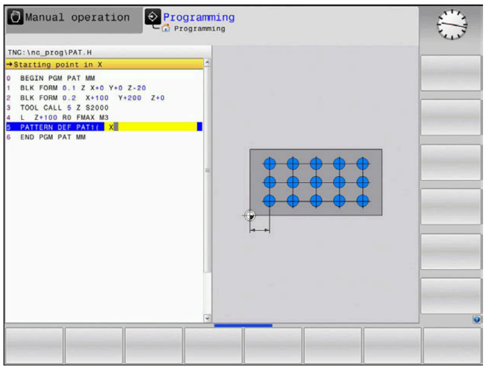
The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.



- **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 principal 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 secondary 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

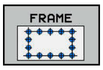
Example

```
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 an individual frame

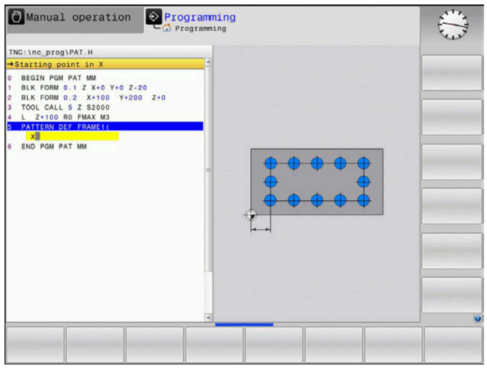
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.
>
> The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position 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 principal 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 secondary 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

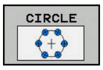
Example

```
10 L  Z+100 R0  FMAX
11 PATTERN DEF  FRAME1
   (X+25  Y+33,5 DX+8  DY+10 NUMX5
   NUMY4 ROT+0  ROTX+0 ROTY+0  Z
   +0)
```



Defining a full circle

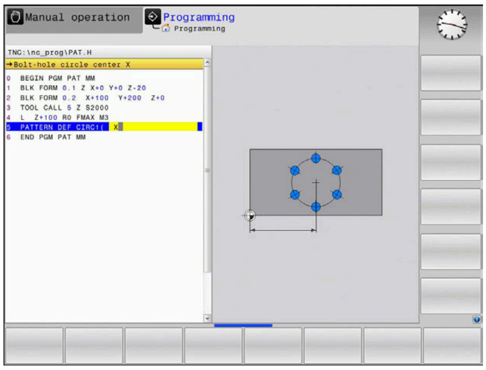
> **i** If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **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

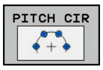
Example

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



Defining a pitch circle

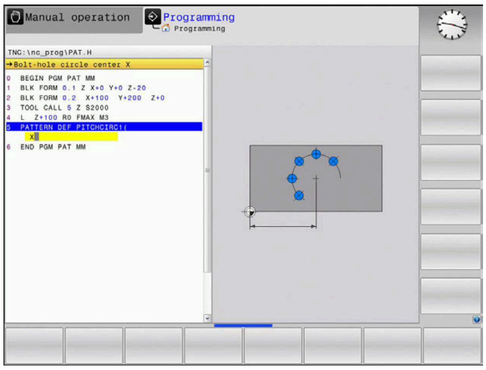
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



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

Example

```
10 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

Proceed as follows:

- Press the **PROGRAMMING** key
- Press the **PGM MGT** key
- > The control opens the file manager.
- Select the folder in which you wish to create the new file
- Enter the name and file type (**.PNT**)
- Press the **ENT** key
- Press the **MM** or **INCH** soft key.
- > The control changes to the program window and displays an empty point 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.

> If you intend to use the point table in SQL queries later, the table name must begin with a letter.
>
> Use the **SORT/ HIDE COLUMNS** soft key 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.

Proceed as follows:

- Use the **CURSOR KEYS** to select the desired point in the table
- Select the **FADE** column
- To activate the hiding option for the point, press the **ENT** key
- To deactivate the hiding option for the point, press the **NO ENT** key

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.

Proceed as follows:

- Press the **PGM CALL** key
- Press the **SELECT POINT TABLE** soft key
- Press the **SELECT FILE** soft key
- Select the point table
- Press the **OK** soft key

If the point table is not stored in the same directory as the NC program, you must enter the complete path.

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**:

Proceed as follows:

- Press the **CYCL CALL** key

- Press the **CYCL CALL PAT** soft key
- Enter a feed rate
- > The control will use this feed rate to traverse between the points.
- Alternatively, press the **F MAX** soft key
- > No input: the control will use the last programmed feed rate.
- Enter a miscellaneous function (M function) if required
- Confirm your input with the **END** key

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

Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function (found under **SPEC FCT**/PROGRAM DEFAULTS) with **Q352**=1. If you do so, the control will always position the tool at the 2nd set-up clearance defined in the cycle.

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

Effect of the point table with 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.

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.

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**.

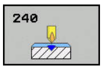
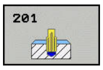
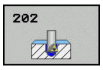
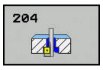
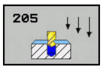
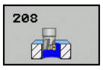
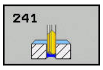
4

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-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth	111
	200 DRILLING With automatic pre-positioning, 2nd set-up clearance	75
	201 REAMING With automatic pre-positioning, 2nd set-up clearance	77
	202 BORING With automatic pre-positioning, 2nd set-up clearance	79
	203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	82
	204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	88
	205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	92
	208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance	100
	241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition	103

4.2 DRILLING (Cycle 200, DIN/ISO: G200)

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 procedure (steps 2 to 4) until the programmed depth is reached (the dwell time from **Q211** is effective with every infeed)
6 Finally, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

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.

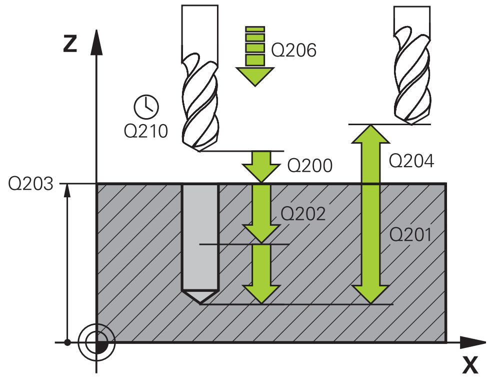
If you want to drill without chip breaking, make sure to define, in the **Q202** parameter, a higher value than the depth **Q201** plus the calculated depth based on the point angle. You can enter a much higher value there.

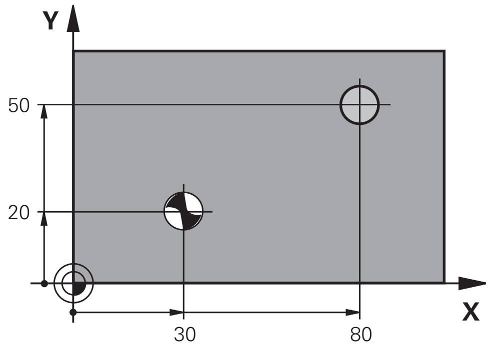
Cycle parameters

- **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 references 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

4.3 REAMING (Cycle 201,DIN/ISO: G201, 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:

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!

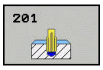
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

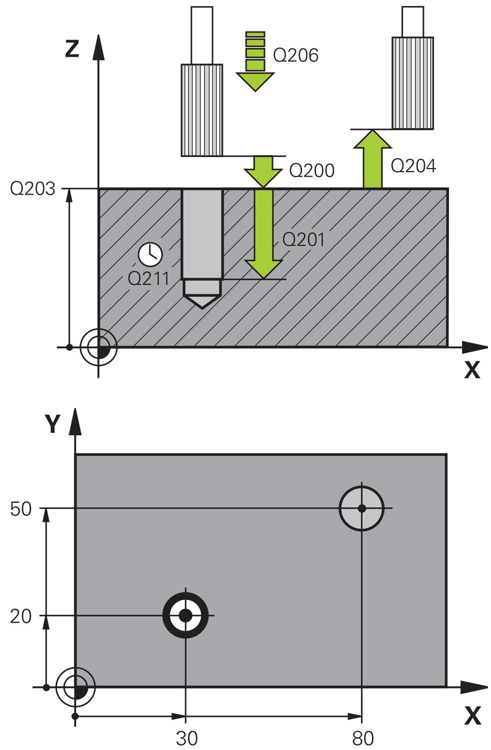
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

Cycle parameters



- **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
```

4.4 BORING (Cycle 202, DIN/ISO: G202, 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:

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
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

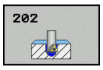
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

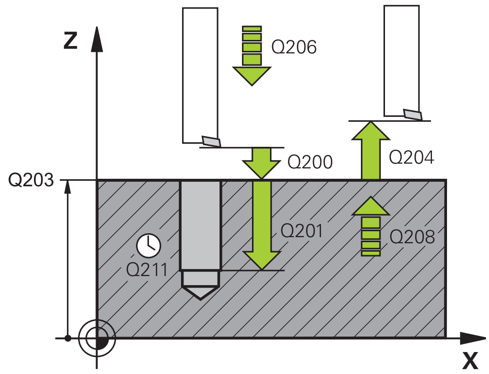
After machining, the control returns the tool to the starting point of the working plane. This way, you can continue positioning the tool incrementally.

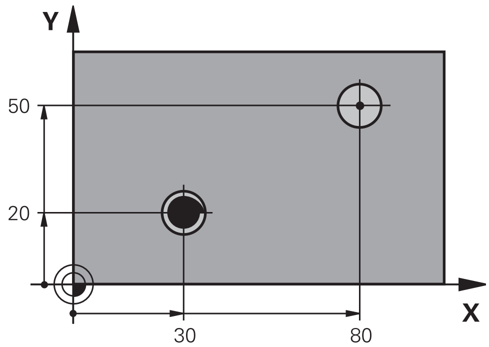
If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

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 principal axis
  2: Disengage the tool in the minus direction of the secondary axis
  3: Disengage the tool in the plus direction of the principal axis
  4: Disengage the tool in the plus direction of the secondary 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
```

4.5 UNIVERSAL DRILLING (Cycle 203, DIN/ISO: G203, 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 DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**
5. When machining without chip breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and remains there for the **DWELL TIME AT TOP Q210**
6. This sequence will be repeated until **depth Q201** is reached.
7. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking 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 the **FEED RATE FOR PLNGNG Q206**
5. The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
6. If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**
7. Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8. Steps 2 to 7 will be repeated until **DEPTH Q201** is reached
9. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE** .The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking 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 the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE** .The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Please note while programming:

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

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

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

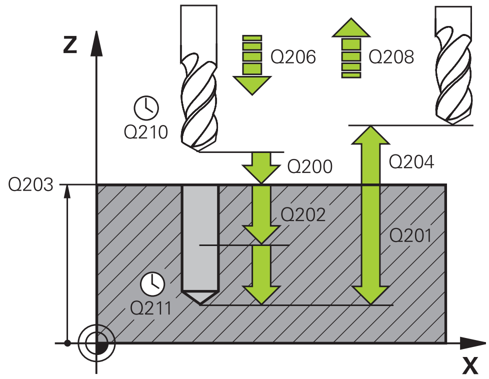
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

- **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

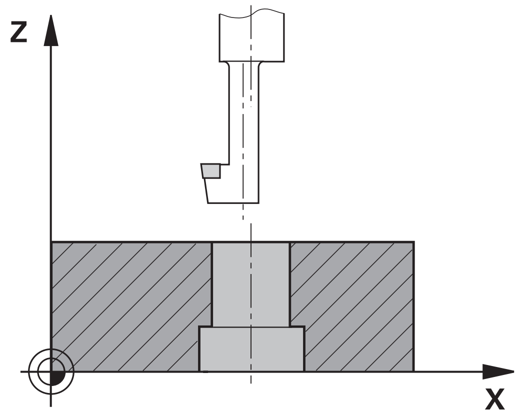
11 CYCL DEF 203 UNIVERSAL DRILLING	
Q200=2	**;SET-UP CLEARANCE**
Q201=-20	**;DEPTH**
Q206=150	**;FEED RATE FOR PLNGNG**
Q202=5	**;PLUNGING DEPTH**
Q211=0	**;DWELL TIME AT TOP**
Q203=+20	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**
Q212=0.2	**;DECREMENT**
Q213=3	**;NR OF BREAKS**
Q205=3	**;MIN. 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**
12 L X+30 Y+20 FMAX M3	
13 CYCL CALL	

- **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 references 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

4.6 BACK BORING (Cycle 204, DIN/ISO: G204, 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:

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
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

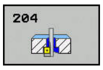
The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

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

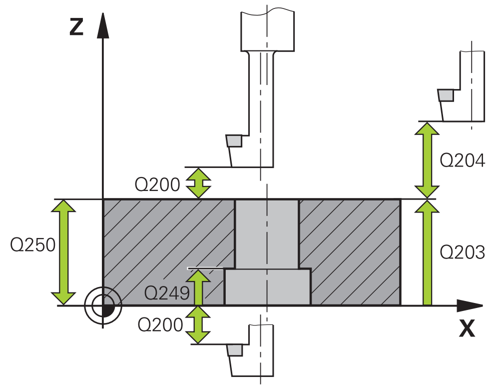
When calculating the starting point for boring, the control considers the cutting edge length of the boring bar and the thickness of the material.

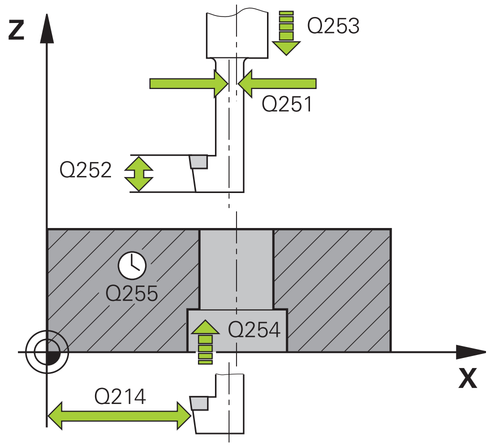
If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.

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 countersinking. Input range: 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Q255 Dwell time in secs.?**: Dwell time in seconds at the top of the bore hole. Input range: 0 to 3600.000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999





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 displaces the tool by the off-center distance (after spindle orientation); programming 0 is not allowed
1: Retract the tool in negative direction of the principal axis
2: Retract the tool in negative direction of the secondary axis
3: Retract the tool in positive direction of the principal axis
4: Retract the tool in positive direction of the secondary 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

4.7 UNIVERSAL PECKING (Cycle 205, DIN/ISO: G205, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

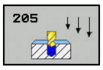
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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.

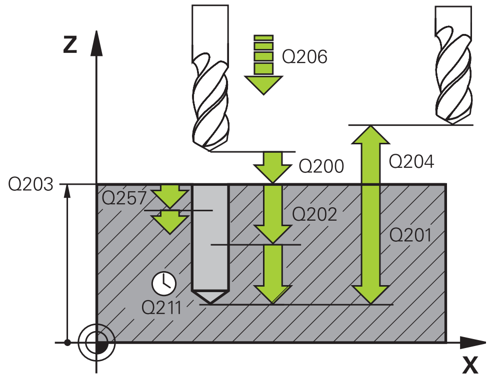
If you use **Q379** to enter a recessed starting point, the control will change the starting point of the infeed movement. Retraction movements are not changed by the control, they are always calculated with respect to the coordinate of the workpiece surface.

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 references 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, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (**Q200**=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (**Q200**=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

Chip removal is at a certain value above the 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 examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with recessed starting point

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

4.8 BORE MILLING (Cycle 208, DIN/ISO: G208, option 19)

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance **Q200** above the workpiece surface
2. In the next step, the control moves in a semi-circle to approach the helical path (starting at the center)
3. The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate **F**.
4. When the drilling depth is reached, the control once again traverses a full circle to remove the material remaining after the initial plunge.
5. The control then positions the tool at the center of the hole again, retracting it to the set-up clearance **Q200**.
6. This procedure is repeated until the nominal diameter is reached (the control calculates the stepover by itself)
7. Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance **Q204** at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

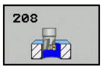
Note that if the infeed distance is too large, the tool or the workpiece may be damaged.

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.

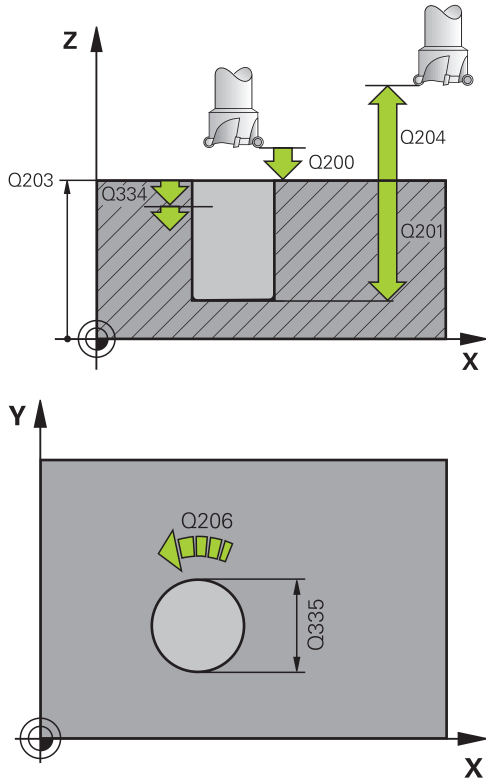
When calculating the infeed and the overlap factor, the control takes the corner radius DR2 of the current tool into account.

For the first helical path, the overlap factor is set as high as possible to prevent the tool from touching the ground of the hole. All other paths are distributed uniformly.

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): Enter the dimension of the predrilled diameter. Input range: 0 to 99999.9999
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (if you enter 0, climb milling is performed)



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

4.9 SINGLE-LIP DEEP HOLE DRILLING (Cycle 241, DIN/ISO: G241, 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 and set the speed to 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:

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!

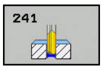
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

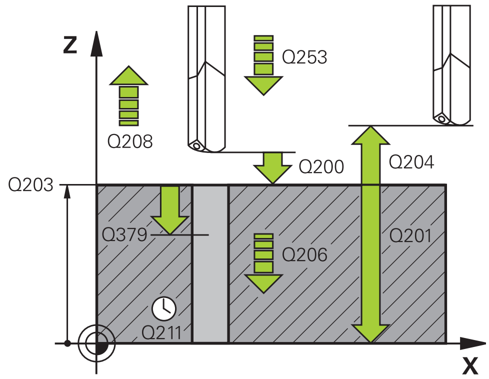
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

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

Cycle parameters



- **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



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
```

- **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
- **Q429 M function for coolant on?**: Miscellaneous function for switching on the coolant. The control switches the coolant on if the tool is in the hole at **Q379 STARTING POINT**. Input range: 0 to 999
- **Q430 M function for coolant off?**: Miscellaneous function for switching off the coolant. The 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, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (**Q200**=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (**Q200**=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

Chip removal is at a certain value above the 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 examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with recessed starting point

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

4.10 CENTERING (Cycle 240, DIN/ISO: G240, 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:

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!

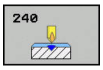
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

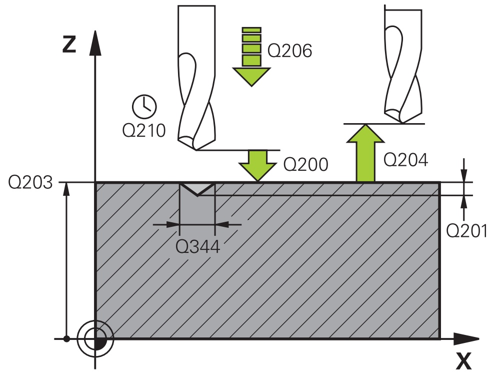
Program a positioning block for the starting point (hole center) in the working plane with the radius compensation **R0**

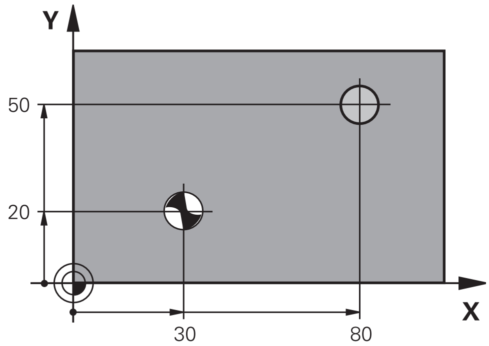
The algebraic sign for the **Q344** (diameter) or **Q201** (depth) cycle parameter determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

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



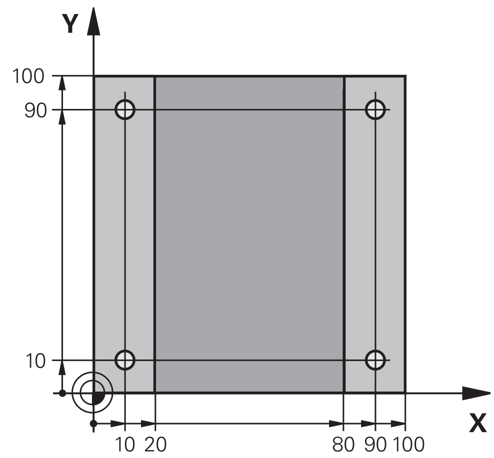


Example

```
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
```

4.11 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	

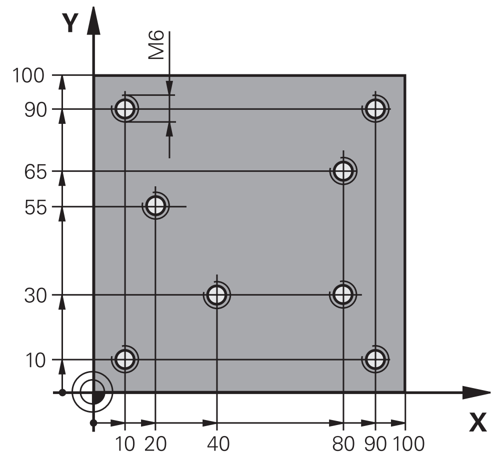
Example: Using drilling cycles in connection with PATTERN DEF

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 118



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	

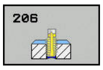
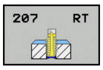
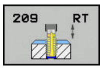
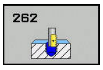
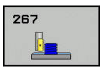
5

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 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	119
	207 TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	122
	209 TAPPING WITH CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	126
	262 THREAD MILLING Cycle for milling a thread in pre-drilled material	133
	263 THREAD MILLING/ COUNTERSINKING Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	137
	264 THREAD DRILLING/ MILLING Cycle for drilling into solid material with subsequent milling of the thread with a tool	141
	265 HELICAL THREAD DRILLING/MILLING Cycle for milling the thread into solid material	145
	267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer	149

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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

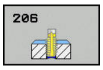
For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.

In Cycle 206, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.

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

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

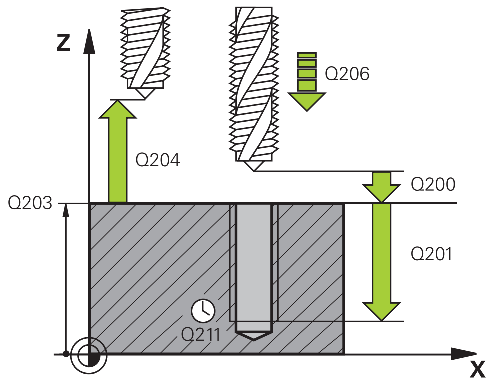
Cycle parameters



- **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!

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

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

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.

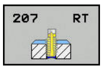
For tapping, the spindle and the tool axis are always synchronized with each other. The synchronization can be carried out while the spindle is rotating or while it is stationary.

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.

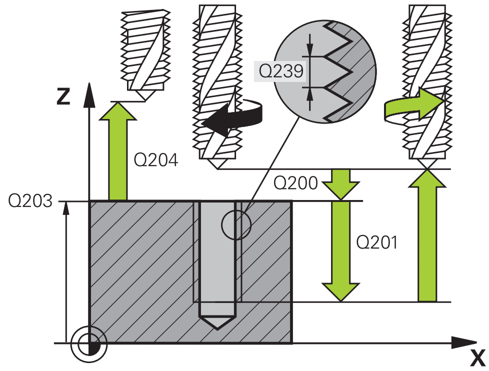
Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

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

Cycle parameters



- **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

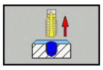
26 CYCL DEF 207 RIGID TAPPING NEU
Q200=2 ;SET-UP CLEARANCE
Q201=-20 ;DEPTH OF THREAD
Q239=+1 ;THREAD PITCH
Q203=+25 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE

Retracting after a program interruption

Retracting in the Positioning with Manual Data Input operating mode

Proceed as follows:

- To interrupt thread cutting, press the **NC stop** key



- Press the retract soft key.

- Press **NC start**
- \> The tool retracts from the hole and moves to the starting point of machining. The spindle is stopped automatically. The control displays a message.

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

Proceed as follows:

- To interrupt the program, press the **NC stop** key

- Press the **MANUAL TRAVERSE** soft key
- Retract the tool in the active spindle axis

- To continue program execution, press the **RESTORE POSITION** soft key

- Then press **NC start**
- \> The control returns the tool to the position it had assumed before the **NC stop** key was pressed.

NOTICE

Danger of collision!

If you 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, DIN/ISO: G209, 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:

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

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

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.

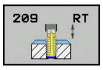
For tapping, the spindle and the tool axis are always synchronized with each other. Synchronization can take place while the spindle is stationary.

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

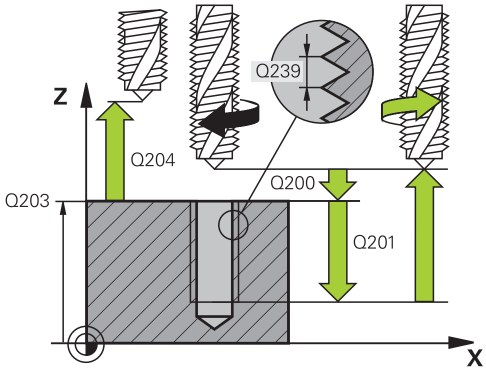
Using the **CfgThreadSpindle** parameter (no. 113600), you can set the following:

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

Cycle parameters



- **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

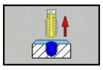
26 CYCL DEF 209 TAPPING W/ CHIP BRKG	
Q200=2	**;SET-UP CLEARANCE**
Q201=-20	**;DEPTH OF THREAD**
Q239=+1	**;THREAD PITCH**
Q203=+25	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**
Q257=5	**;DEPTH FOR CHIP BRKNG**
Q256=+1	**;DIST FOR CHIP BRKNG**
Q336=50	**;ANGLE OF SPINDLE**
Q403=1.5	**;RPM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

Retracting after a program interruption

Retracting in the Positioning with Manual Data Input operating mode

Proceed as follows:

- To interrupt thread cutting, press the **NC stop** key



- Press the retract soft key.

- Press **NC start**
- > The tool retracts from the hole and moves to the starting point of machining. The spindle is stopped automatically. The control displays a message.

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

Proceed as follows:

- To interrupt the program, press the **NC stop** key

- Press the **MANUAL TRAVERSE** soft key
- Retract the tool in the active spindle axis

- To continue program execution, press the **RESTORE POSITION** soft key

- Then press **NC start**
- > The control returns the tool to the position it had assumed before the **NC stop** key was pressed.

NOTICE

Danger of collision!

If you 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 can set the compensation in **TOOL CALL** using the **DR** delta radius).
- Cycles 262, 263, 264, and 267 can only be used with clockwise rotating tools, cycle 265 is suitable for clockwise or counterclockwise rotating tools
- The working direction is determined by the following input parameters: Algebraic sign **Q239** (+ = right-hand thread / – = left-hand thread) and type of milling **Q351** (+1 = climb / –1 = up-cut).

  The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/ Up-cut	Work direction
Right-handed	+	+1(RL)	Z+
Left-handed	–	–1(RR)	Z+
Right-handed	+	–1(RR)	Z–
Left-handed	–	+1(RL)	Z–

External thread	Pitch	Climb/ Up-cut	Work direction
Right-handed	+	+1(RL)	Z–
Left-handed	–	–1(RR)	Z–
Right-handed	+	–1(RR)	Z+
Left-handed	–	+1(RL)	Z+

NOTICE

Danger of collision!

If you program the plunging depth values with different algebraic signs a collision may occur.

- Make sure to program all depth values with the same algebraic sign. Example: If you program the **Q356** COUNTERSINKING DEPTH parameter with a negative sign, then **Q201** DEPTH OF THREAD must also have a negative sign
- If you 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.

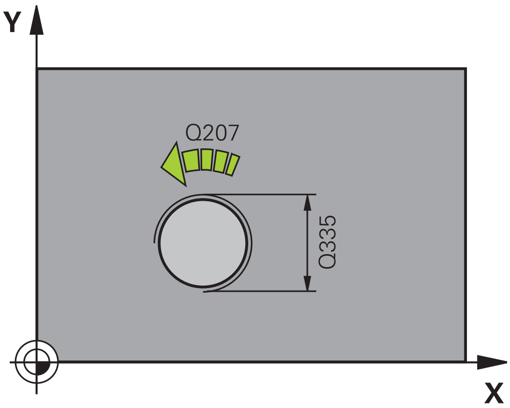
- Stop the program run if the tool breaks
- Change to Positioning with Manual Data Input mode of operation
- First move the tool in a linear movement towards the hole center
- Retract the tool in the tool axis direction

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, DIN/ISO: G262, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

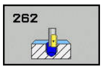
If you program the thread depth =0, the cycle will not be executed.

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 tool diameter is smaller than the nominal thread diameter by four times the thread pitch.

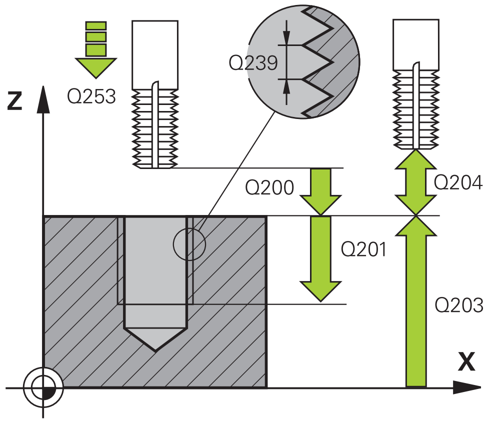
Note that the control makes a compensating movement in the tool axis before the approach. The length of the compensation movement is at most half of the thread pitch. Ensure sufficient space in the hole!

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

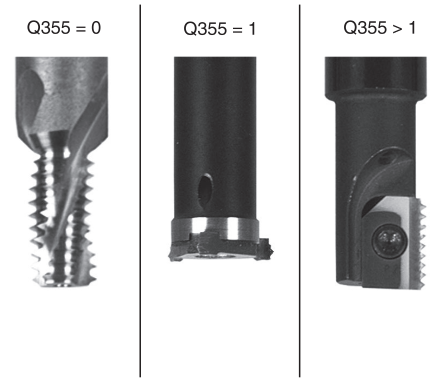
Cycle parameters



- **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. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (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**



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 MILLING
Q512=0	;FEED FOR APPROACH

5.7 THREAD MILLING/COUNTERSINKING (Cycle 263, DIN/ISO: G263, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

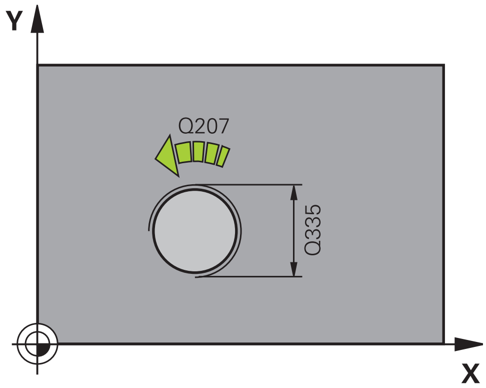
The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1. Thread depth
2. Countersinking depth
3. Depth at front

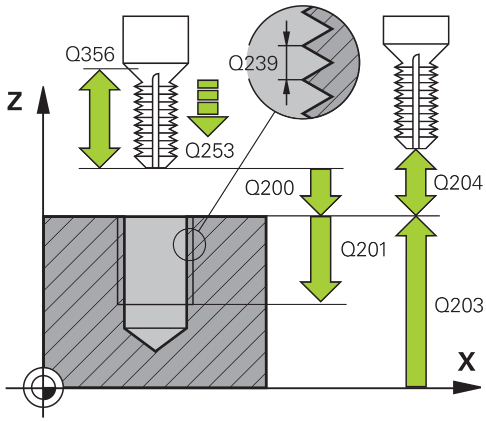
If you program one of the depth parameter to be 0, the control does not execute that step.

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.

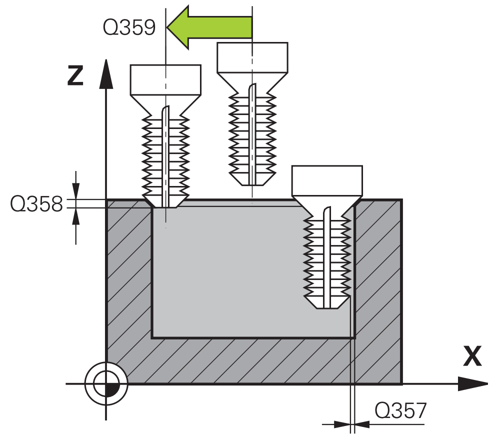
Cycle parameters





- **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. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (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 countersinking. 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 MILLING
Q512=0	;FEED FOR APPROACH

5.8 THREAD MILLING (Cycle 264, DIN/ISO: G264, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

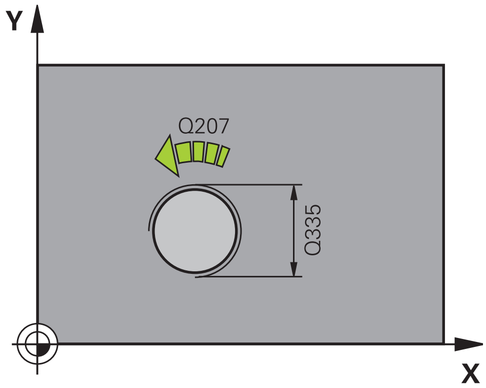
Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

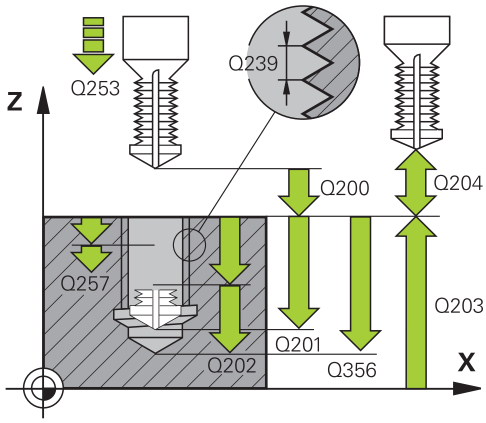
Cycle parameters

- **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. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (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 MILLING
Q512=0	;FEED FOR APPROACH

5.9 HELICAL THREAD DRILLING/MILLING (Cycle 265, DIN/ISO: G265, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

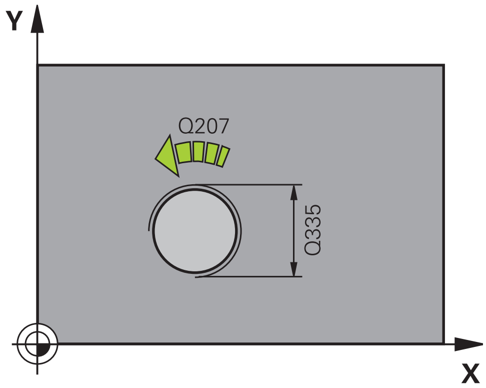
If you program one of the depth parameter to be 0, the control does not execute that step.

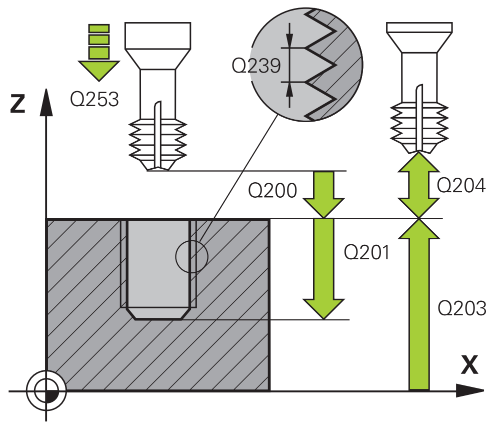
If you change the thread depth, the control will automatically move the starting point for the helical movement.

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

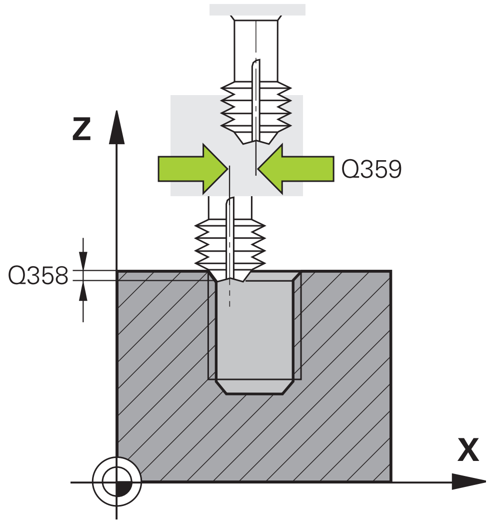
Cycle parameters

- **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 countersinking. 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 MILLING

5.10 OUTSIDE THREAD MILLING (Cycle 267, DIN/ISO: G267, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

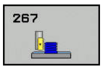
The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

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

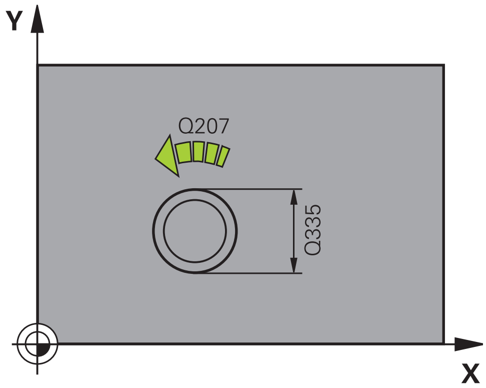
If you program one of the depth parameter to be 0, the control does not execute that step.

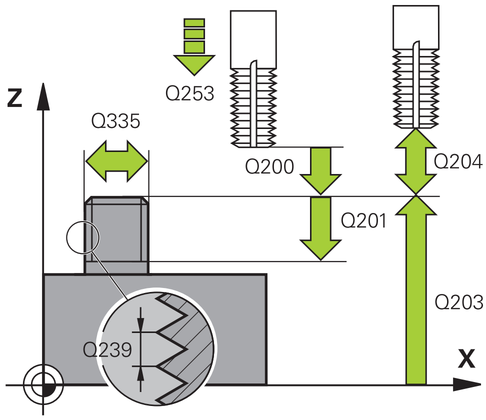
The algebraic sign for the cycle parameter "thread depth" determines the working direction.

Cycle parameters

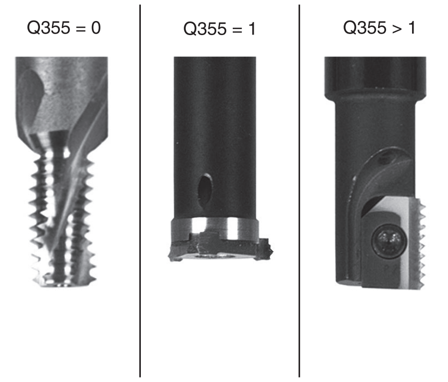


- **Q335 Nominal diameter?**: Nominal thread diameter. Input range: 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of 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. The direction of spindle rotation is taken into account.
  +1 = Climb milling
  –1 = Up-cut milling (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 countersinking. 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 267  OUTSIDE THREAD
   MLLNG
   Q335=10     ;NOMINAL DIAMETER
   Q239=+1.5   ;THREAD PITCH
   Q201=-20    ;DEPTH OF THREAD
   Q355=0      ;THREADS PER STEP
   Q253=750    ;F PRE-POSITIONING
   Q351=+1     ;CLIMB OR UP-CUT
   Q200=2      ;SET-UP CLEARANCE
   Q358=+0     ;DEPTH AT FRONT
   Q359=+0     ;OFFSET AT FRONT
   Q203=+30    ;SURFACE COORDINATE
   Q204=50     ;2ND SET-UP CLEARANCE
   Q254=150    ;F COUNTERBORING
   Q207=500    ;FEED RATE MILLING
   Q512=0      ;FEED FOR APPROACH
```

5.11 Programming Examples

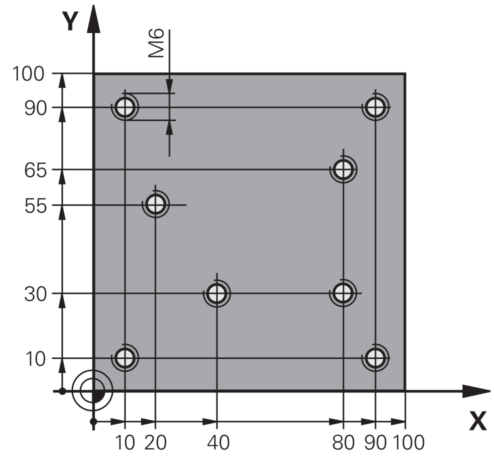
Example: Thread milling

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
NRXYZ
0 +10 +10 +0
1 +40 +30 +0
2 +90 +10 +0
3 +80 +30 +0
4 +80 +65 +0
5 +90 +90 +0
6 +10 +90 +0
7 +20 +55 +0
[END]

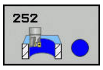
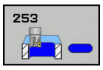
6

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
	258 POLYGON STUD Roughing/finishing cycle for machining a regular polygon	190
	233 FACE MILLING Machining the face with up to three limits	196

6.2 RECTANGULAR POCKET (Cycle 251, DIN/ISO: G251, 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 (**Q370**) and the finishing allowance (**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!

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, 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.

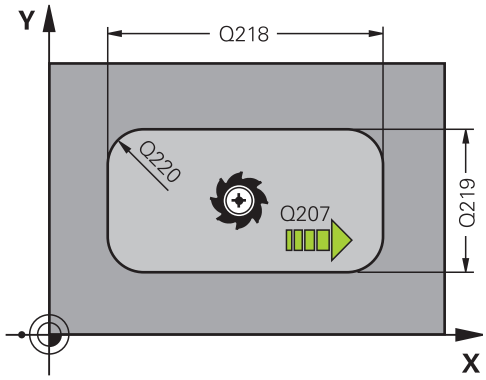
Program a sufficient set-up clearance so that the tool cannot jam because of chips.

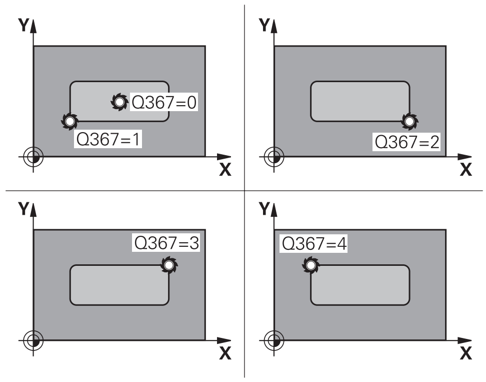
The control outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter (no. 201006).

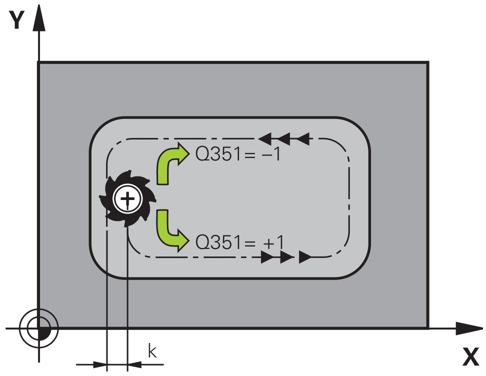
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

Cycle parameters

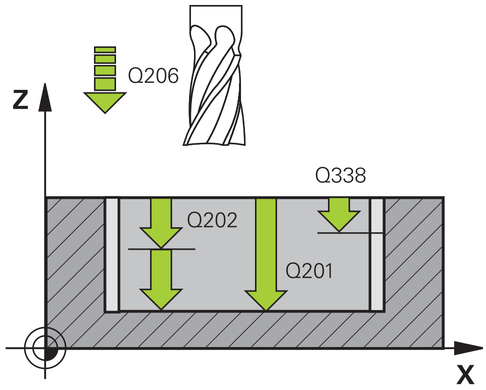
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q218 First side length?** (incremental): Pocket length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q219 Second side length?** (incremental): Pocket length, parallel to the secondary 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. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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

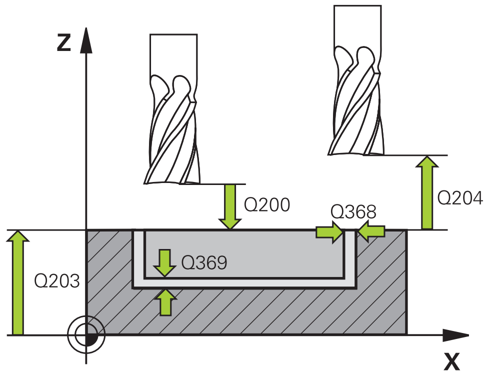






- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool 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 with a single 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**
- **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: 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





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
Q207=500	;FEED RATE MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
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	

- **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

6.3 CIRCULAR POCKET (Cycle 252, DIN/ISO: G252, 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 (**Q370**) and the finishing allowance (**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 second set-up clearance **Q204** in the tool axis and returns at rapid traverse to the pocket center.

Finishing

1. If finishing allowances have been defined, the control first finishes the pocket walls, in multiple infeeds, if so specified.
2. The control positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200**
3. The control roughs out the pocket from the inside out, until the diameter **Q223** is reached
4. Then, the control again positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200** and repeats the finishing procedure for the side wall at the new depth
5. The control repeats this process until the programmed diameter is reached
6. After machining to the diameter **Q223**, the control retracts the tool tangentially by the finishing allowance **Q368** plus the set-up clearance **Q200** in the working plane, then retracts it at rapid traverse to 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.

Please note while programming:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.

Pre-position the tool in the 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.

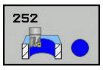
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

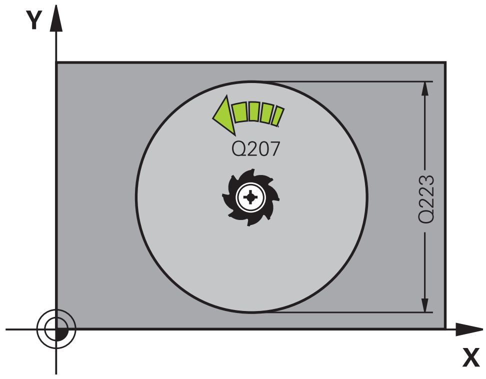
Program a sufficient set-up clearance so that the tool cannot jam because of chips.

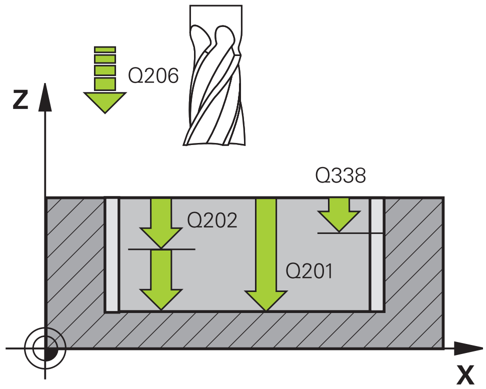
The control outputs an error message during helical plunging if the internally calculated diameter of the helix is smaller than twice the tool diameter. If you are using a center-cut tool, you can switch off this monitoring function via the **suppressPlungeErr** machine parameter (no. 201006).

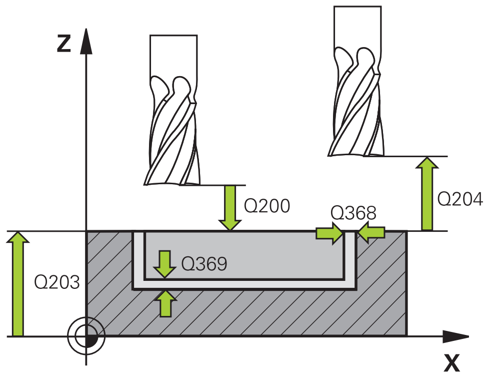
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

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. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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 with a single 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 MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
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	

- **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, 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 (**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.

Please note while programming:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.

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

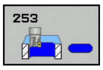
The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

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

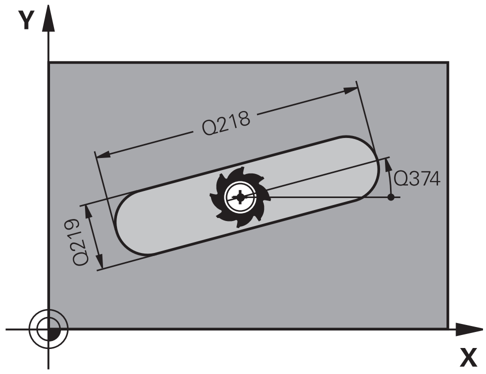
If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

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.

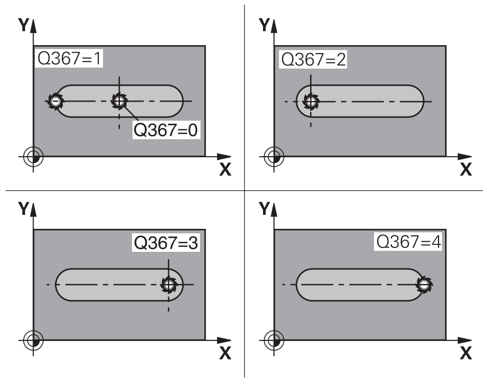
Cycle parameters



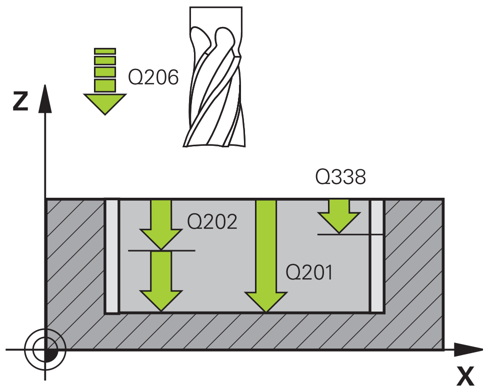
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed



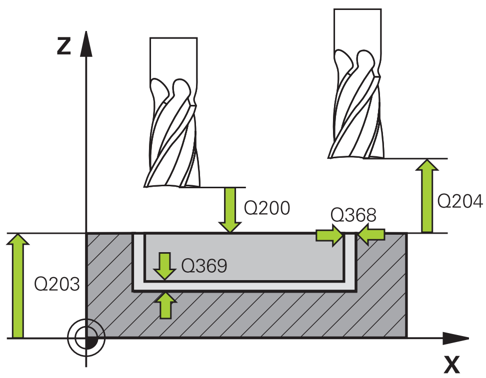
- **Q218 Length of slot?** (value parallel to the principal axis of the working plane): Enter the length of the slot. Input range: 0 to 99999.9999
- **Q219 Width of slot?** (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool 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



- **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. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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 with a single 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 MILLING
Q351=+1	;CLIMB OR UP-CUT

- **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

```
    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, DIN/ISO: G254, 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 (**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:

NOTICE

Danger of collision!

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

- Do not program any incremental dimensions after this cycle
- Program an absolute position in all main axes after this cycle

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.

Pre-position the tool in the 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.

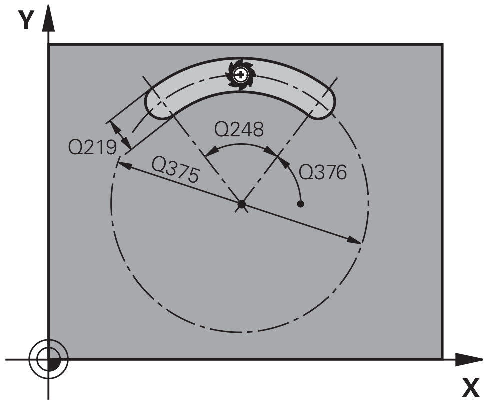
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

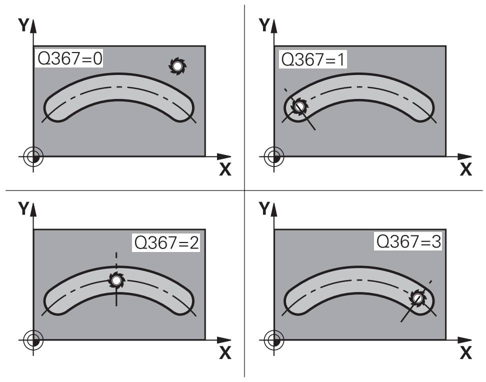
If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.

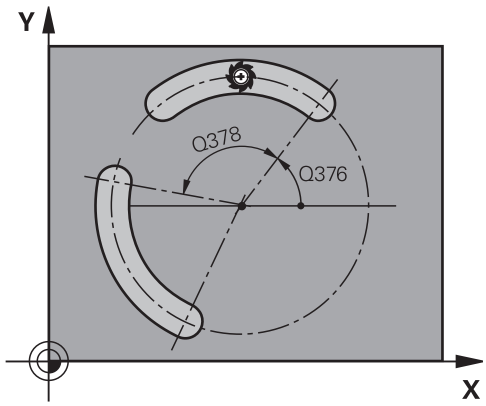
The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

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.

Cycle parameters

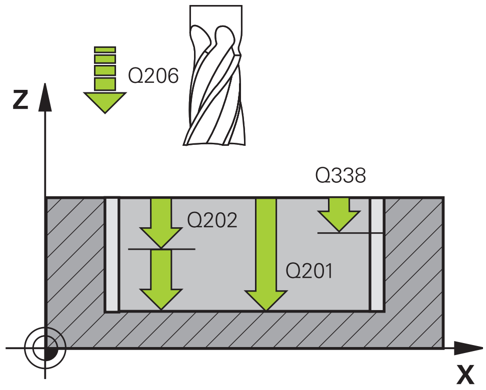


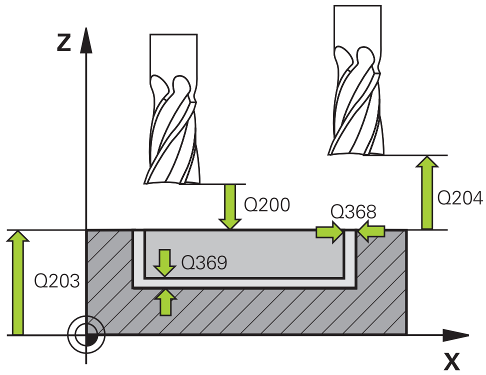




- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (**Q368**, **Q369**) has been programmed
- **Q219 Width of slot?** (value parallel to the secondary 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 principal axis of the working plane. **Only effective if Q367 = 0**. Input range: –99999.9999 to 99999.9999

- **Q217 Center in 2nd axis?** (absolute): Center of the pitch circle in the secondary 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. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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 with a single 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 MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99	

6.6 RECTANGULAR STUD (Cycle 256, DIN/ISO: G256, 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:

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

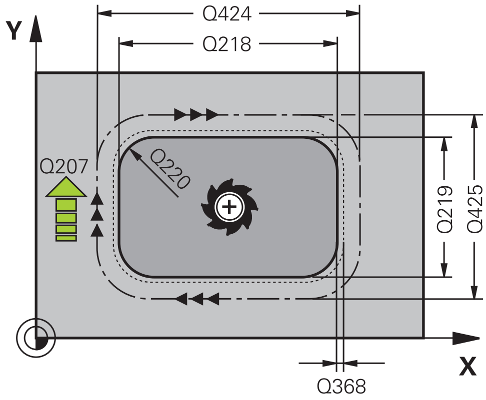
The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

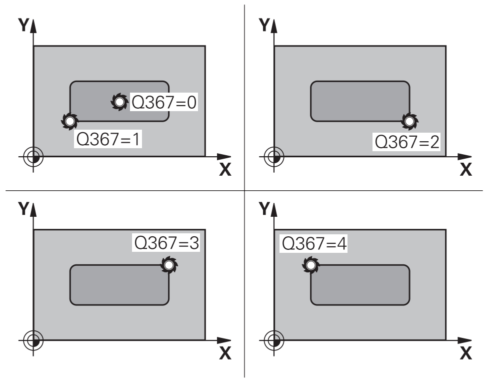
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

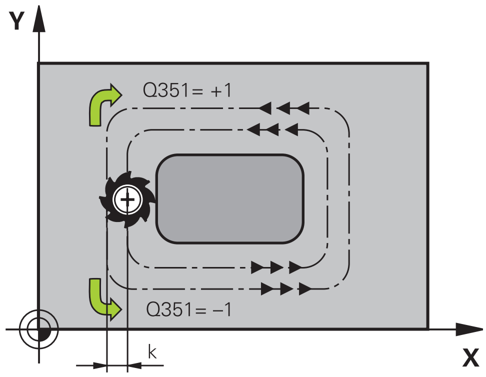
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

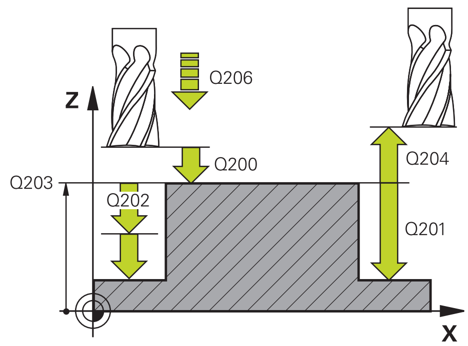
Cycle parameters

- **Q218 First side length?**: Stud length, parallel to the principal 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 principal 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 secondary axis of the working plane. Enter **Workpiece blank side length 2** greater than **Second side length**. The control performs multiple lateral stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q425 Workpiece blank side length 2?**: Length of the stud blank, parallel to the secondary 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. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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 MILLING
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	

- **Q437 Starting position (0...4)?**: Define the approach strategy of the tool:
  0: to the right of the stud (default setting)
  1: lower left corner
  2: lower right corner
  3: upper right corner
  4: upper left corner.
  If approach marks appear on the stud surface during approach with the setting **Q437**=0, then choose another approach position.
- **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 with a single 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, DIN/ISO: G257, 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.

Please note while programming:

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 exact starting position, enter a starting angle between 0° and 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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

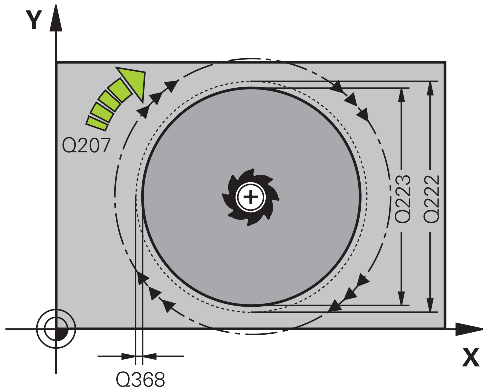
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

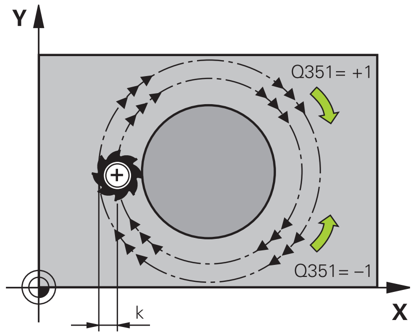
At the end of the cycle, the control returns the tool to the starting position.

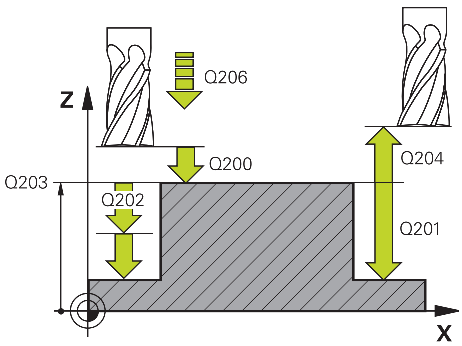
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

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. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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 with a single 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 MILLING
    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, DIN/ISO: G258, 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 other things, on the diameter of the workpiece blank and the angle of rotation of the stud. The angle of rotation is determined with parameter **Q224**
3. The tool moves at rapid traverse **FMAX** to the setup clearance **Q200** and from there with the feed rate for plunging to the first plunging depth.
4. The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5. The control moves the tool on a tangential path from the outside to the inside
6. The tool will be lifted in the direction of the spindle axis to 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:

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.

- Use **Q224** to specify which angle is used to machine the first corner of the polygon stud. Input range: –360° to +360°
- Depending on the angle of rotation **Q224**, the following amount of space must be left next to the stud: At least tool diameter +2 mm

NOTICE

Danger of collision!

At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle need not be the same as the starting position.

- 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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

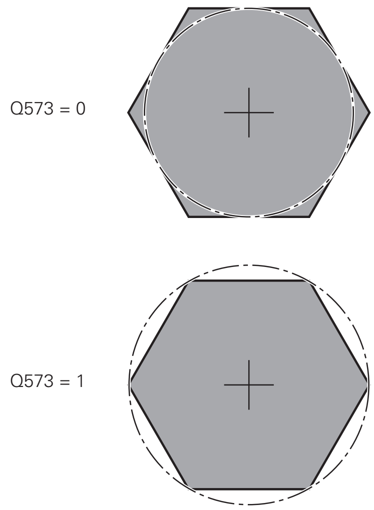
The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

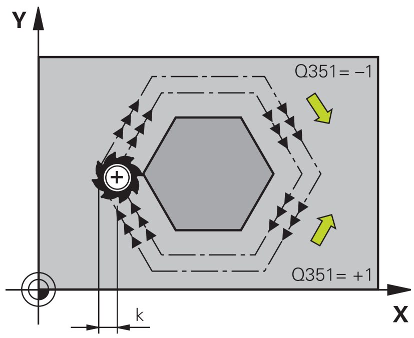
The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

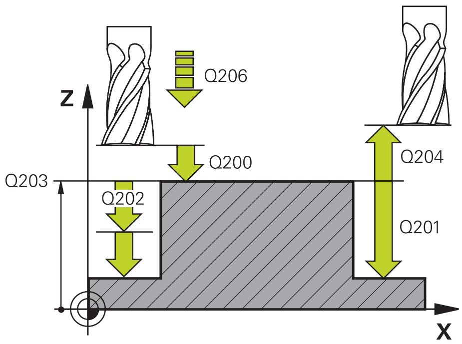
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

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. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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**

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 MILLING
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	

- **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 with a single 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, DIN/ISO: G233, option 19)

Cycle run

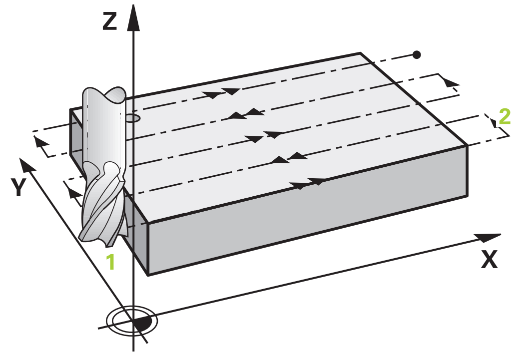
With Cycle 233, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

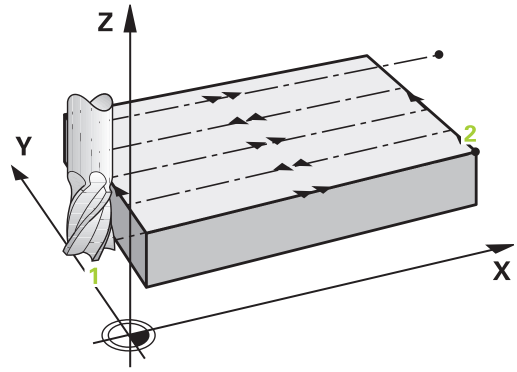
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover when retracting at rapid traverse
- **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

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.

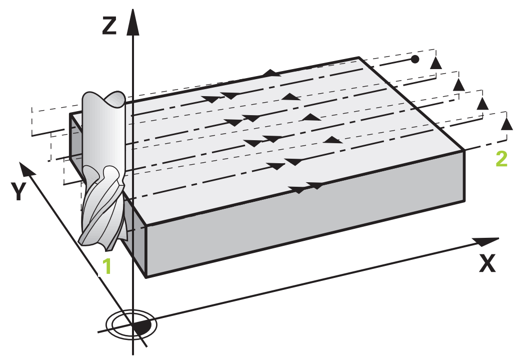


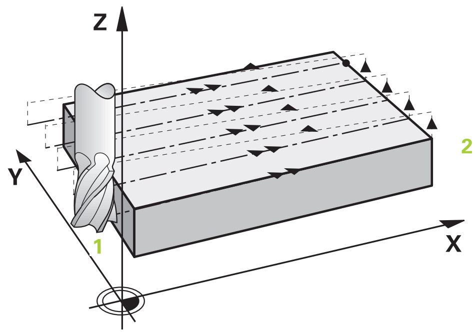


4 The control moves the tool to end point **2** at the programmed feed rate for milling
5 Then the control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and 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

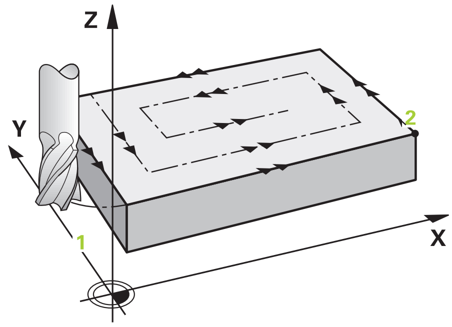
The strategies **Q389**=2 and **Q389**=3 differ in the overtravel during face milling. If **Q389**=2, the end point lies outside of the surface, with **Q389**=3, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389**=2 is used, the control additionally moves the tool beyond the level surface by the tool radius.





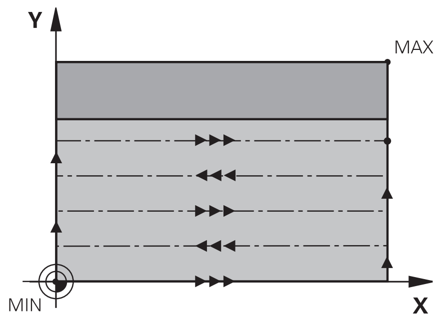
4 The tool subsequently advances at the programmed feed rate for milling to end point **2**
5 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves at **FMAX** directly back to the starting point in the next pass. The control calculates 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**.

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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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).

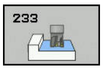
The control reduces the plunging depth to the LCUTS cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.

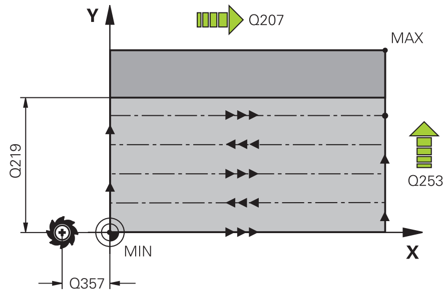
If you define **Q370 TOOL PATH OVERLAP** >1, the programmed overlap factor will be taken into account right from the first machining path.

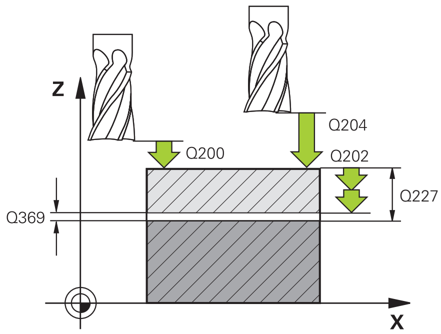
Cycle 233 monitors the entries made for the tool or cutting edge length in **LCUTS** in the tool table. If the tool or cutting edge length is not sufficient for a finishing operation, the control will subdivide the process into multiple machining steps.

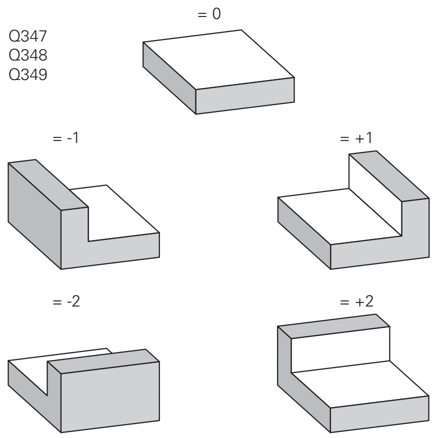
If a limit (**Q347**, **Q348** or **Q349**) was programmed in the machining direction **Q350**, the cycle will extend the contour in the infeed direction by corner radius **Q220**. The specified surface will be machined completely.

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 being machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate outside the surface being machined
  3: Line-by-line machining, retraction and stepover at the positioning feed rate at the edge of the surface being 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: Principal axis = machining direction
  2: Secondary axis = machining direction
- **Q218 First side length?** (incremental): Length of the surface to be machined in the principal 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 secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referencing 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** influences the following situations:
  Approaching the first plunging depth: **Q357** is the lateral distance from the tool to the workpiece
  Roughing with the milling strategies Q389=0 to 3: The surface to be machined in the **Q350 MILLING DIRECTION** is increased by the value from **Q357** if there is no limit set in this direction
  Side finishing: The paths are extended by the value in **Q357** in the **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**

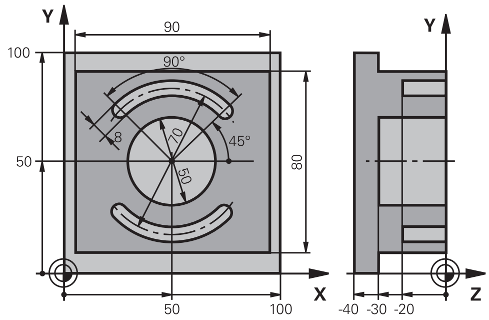
Example

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 MILLING
Q385=500	;FINISHING FEED RATE
Q253=750	;F PRE-POSITIONING
Q357=2	;CLEARANCE TO SIDE
Q200=2	;SET-UP CLEARANCE
Q204=50	;2ND SET-UP CLEARANCE
Q347=0	;1ST LIMIT
Q348=0	;2ND LIMIT
Q349=0	;3RD LIMIT
Q220=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	

- **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 the machining of the plane surface to the corresponding starting point coordinate or side length: (not possible with helical machining):
  Input **0**: No limiting
  Input **–1**: Limit in negative principal axis
  Input **+1**: Limiting in positive principal axis
  Input **–2**: Limiting in negative secondary axis
  Input **+2**: Limiting in positive secondary axis
- **Q348 2nd limit?**: See parameter 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 with a single infeed. Input range: 0 to 99999.9999
- **Q367 Surface position (-1/0/1/2/3/4)?**: Position of the surface with respect to 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 MILLING	
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 MILLING	

Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q366=1 ;PLUNGE	
Q385=750 ;FINISHING FEED RATE	
Q439=0 ;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99	Cycle call for circular pocket
9 TOOL CALL 2 Z S5000	Tool call: slot milling cutter
10 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 MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-20 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q366=1 ;PLUNGE	
Q385=500 ;FINISHING FEED RATE	
Q439=0 ;FEED RATE REFERENCE	
11 CYCL CALL FMAX M3	Cycle call for slots
12 L Z+250 R0 FMAX M2	Retract the tool, end program
13 END PGM C210 MM	

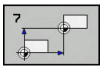
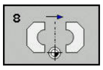
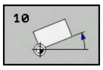
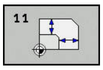
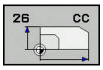
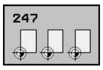
7

Cycles: Coordinate Transformations

7.1 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	209
	8 MIRRORING Mirroring contours	216
	10 ROTATION Rotating contours in the working plane	218
	11 SCALING FACTOR Increasing or reducing the size of contours	220
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	221
	19 WORKING PLANE Machining in a tilted coordinate system on machines with swivel heads and/or rotary tables	223
	247 PRESETTING Presetting during the program run	230

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.

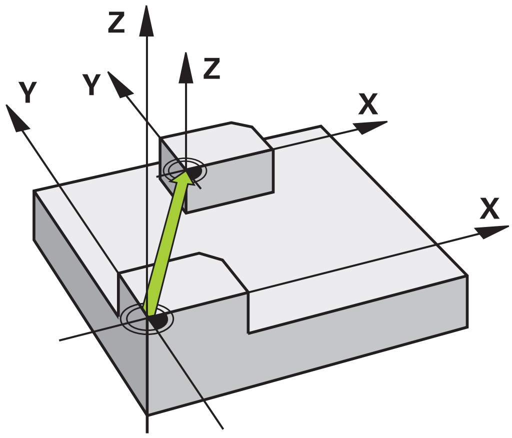
Reset coordinate transformation:

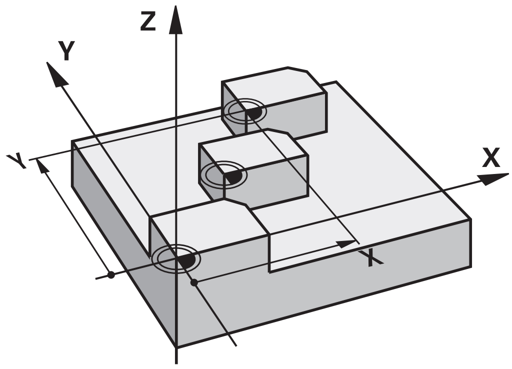
- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM NC block (these M functions depend on the machine parameters)
- Select a new NC program

7.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54)

Effect

> Refer to your machine manual!





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

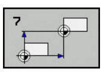
- To shift the datum back to the coordinates X=0, Y=0 etc., program another cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.

Please note while programming

> Possible datum shift values in the rotary axes will be specified by your machine tool builder in the **presetToAlignAxis** parameter (no. 300203).
>
> In the machine parameter **CfgDisplayCoordSys** (no. 127501), the machine tool builder specifies the coordinate system in which the status display shows an active datum shift.

> This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

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

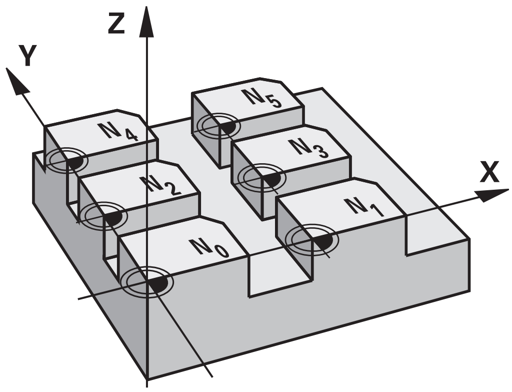
7.3 DATUM SHIFT with datum tables (Cycle 7, DIN/ISO: G53)

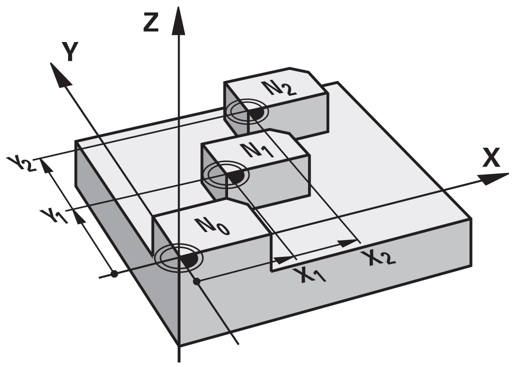
Effect

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:

In the machine parameter **CfgDisplayCoordSys** (no. 127501), the machine tool builder specifies the coordinate system in which the status display shows an active datum shift.

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Datums from a datum table **always and exclusively** reference the current preset.

If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.

If you work without **SEL TABLE,** then you must activate the desired datum table before the test run or the program run (this applies also to the program run):

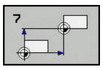
- Use the file manager to select the desired table for a test run in the **Test Run** operating mode: The table now has the status S
- Use the file manager in the **Program run, single block** and **Program run, full sequence** operating modes to select the desired table for program run: The table receives the status M

The coordinate values from datum tables are only effective with absolute coordinate values.

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:

Proceed as follows:

PGM CALL
- Press the **PGM CALL** key

SELECT DATUM TABLE
- Press the **SELECT DATUM TABLE** soft key
- Enter the complete path name of the datum table

SELECT FILE
- Alternative: Press the **SELECT FILE** 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.

Proceed as follows:

PGM MGT
- Press the **PGM MGT** key

SELECT TYPE
- Press the **SELECT TYPE** soft key

SHOW ALL
- Press the **SHOW ALL** soft key
- Select the desired table or enter a new file name
- Select the file with the **ENT** key

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
FIND	Find (a small window opens where you can enter the text or value you are looking for)
RESET TABLE	Reset table
BEGIN LINE	Move the cursor to the beginning of the 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
INSERT LINE	Insert line (only possible at the end of table)
DELETE LINE	Delete line
SORT/ HIDE COLUMNS	Sort or hide columns (a window opens)
MORE FUNCTIONS	Displays more functions: Delete, select, deselect, save as
RESET COLUMN	Reset the column
EDIT CURRENT FIELD	Edit the current field
SORT	Sort the datums (a window opens where you can select the sorting order)

Editing the datum table in the Program Run, Single Block or Program Run, Full Sequence operating mode

Select the datum table in the **Program Run, Full Sequence/Single Block** operating mode.

Proceed as follows:

- ▷ Shift the soft-key row
- SELECT COMPENS. TABLES — Press the **SELECT COMPENS. TABLES** soft key
- DATUM TABLE — Press the **DATUM TABLE** soft key

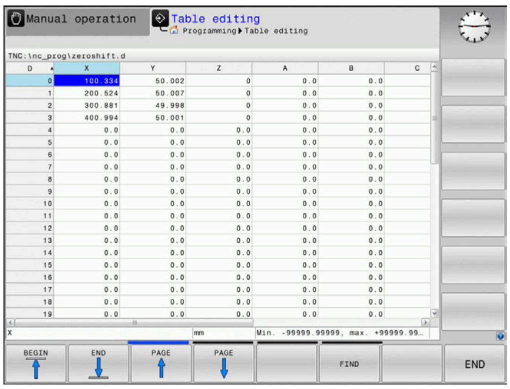
To capture the actual position and store it in the datum table, proceed as follows:

- EDIT OFF ON — Set the **EDIT** soft key to **ON**
- Use the arrow keys to navigate to the desired source
- Press the **ACTUAL POSITION CAPTURE** key
- > The control only captures the actual position for the axis in which the cursor is currently positioned.

> 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.
>
> The change of a datum will only become effective after you have called Cycle 7 again.
>
> While an NC program is running, you cannot access the datum table. For compensations during the program run, use the **COMPENS. TABLE T-CS** or **COMPENS. TABLE WPL-CS** soft key.
> **Further information:** User's Manual for Conversational Programming

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.



> 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.

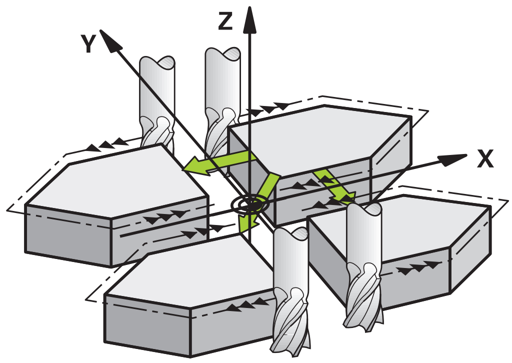
7.4 MIRRORING (Cycle 8, DIN/ISO: G28)

Effect

The control can machine the mirror image of a contour in the working plane.

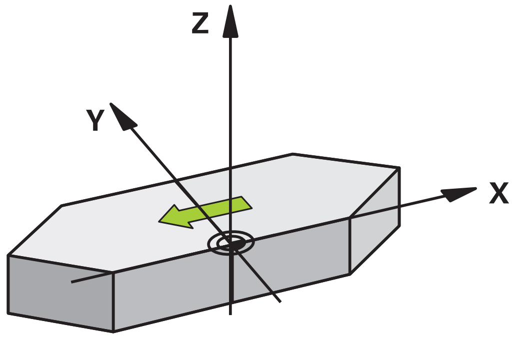
Mirroring becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed; this does not apply to SL cycles
- If you mirror two axes, the machining direction remains the same.



The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.



Resetting

Program the MIRROR IMAGE cycle once again with **NO ENT**.

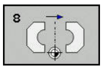
Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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
```

7.5 ROTATION (Cycle 10, DIN/ISO: G73)

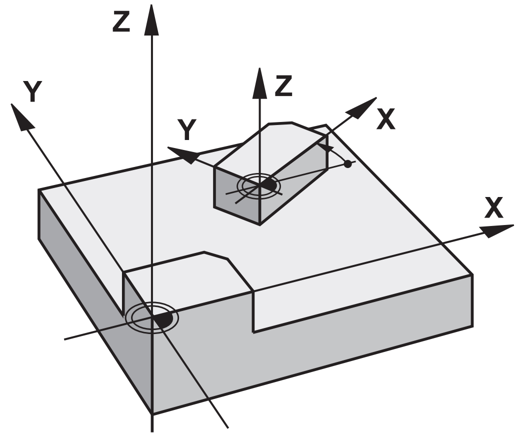
Effect

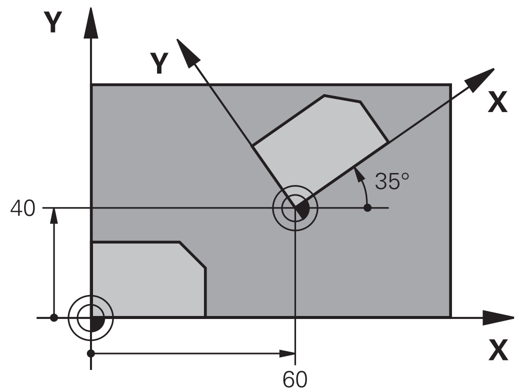
Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

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°.

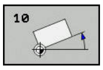
Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle 10 cancels an active radius compensation. If necessary, reprogram the radius compensation.

After defining Cycle 10, 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
```

7.6 SCALING (Cycle 11, DIN/ISO: G72)

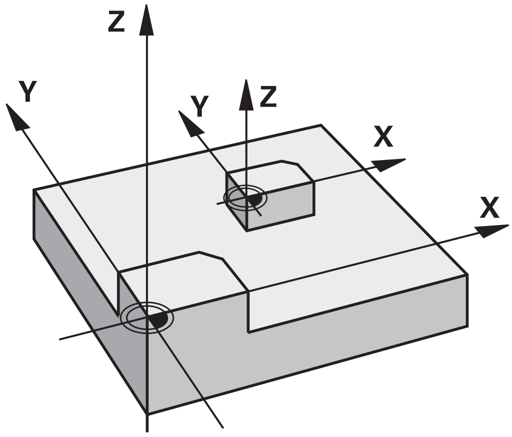
Effect

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

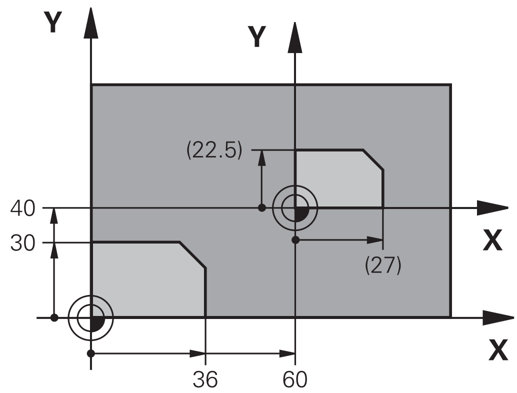


Prerequisite

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)

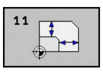


> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

7.7 AXIS-SPECIFIC SCALING (Cycle 26)

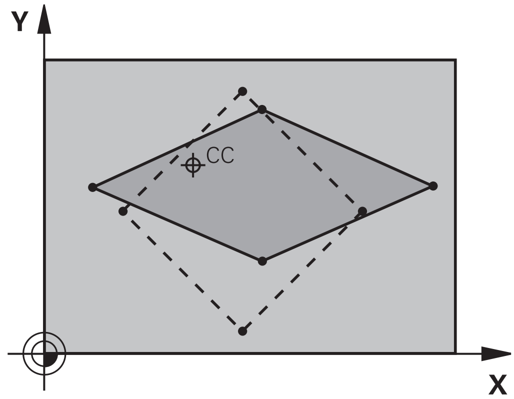
Effect

With Cycle 26, you can account for shrinkage and oversize factors for each axis.

The factor defined for SCALING becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active scaling factor is shown in the additional status display.

Resetting

Program the SCALING cycle once again with a scaling factor of 1 for the corresponding axis.



Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

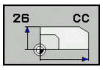
Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

You can program each coordinate axis with its own axis-specific scaling factor.

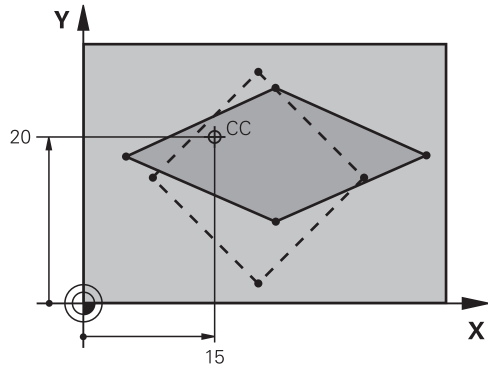
In addition, you can enter the coordinates of a center for all scaling factors.

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

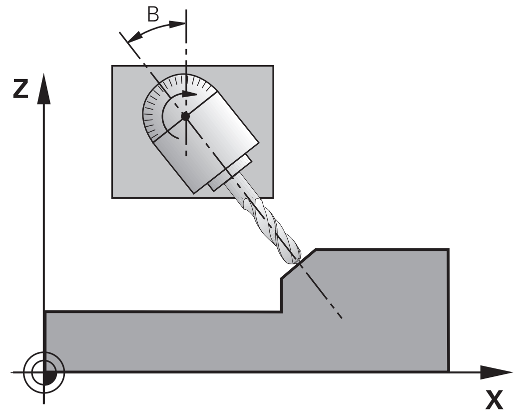
7.8 WORKING PLANE (Cycle 19, DIN/ISO: G80, option 1)

Effect

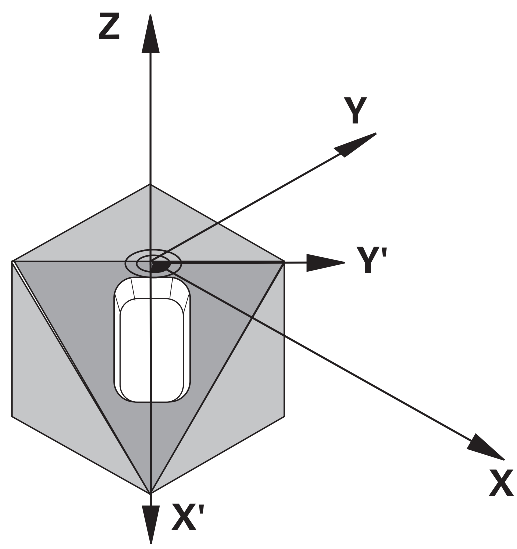
In Cycle 19 you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- 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 angles 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.



> 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 adapted 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).

In the machine parameter **CfgDisplayCoordSys** (no. 127501), the machine tool builder specifies the coordinate system in which the status display shows an active datum shift.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

This cycle can also be used in the **FUNCTION MODE TURN** machining mode if this mode is executed using radial facing slide kinematics.

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.

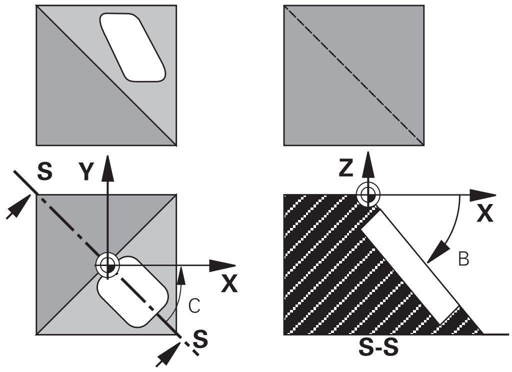
If you use the Cycle 19 while M120 is active, the control automatically cancels the radius compensation, which also cancels the M120 function.

Write the program as if the machining process was to be executed in a non-tilted plane.

If you call the cycle again for other angles, you do not need to reset the machining parameters.

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.

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 have been 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 geometry or 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 **Tilt working plane**

3rd: Activate rotation

...

Workpiece machining

...

1st: Reset rotation

2nd: Reset **Tilt working plane**

3rd: Reset datum shift

Procedure for working with Cycle 19 WORKING PLANE

Proceed as follows:

- Create the NC program
- Clamp the workpiece
- Set any presets
- Start the NC program

Creating the NC program:

- Call the defined tool
- Retract in the spindle axis
- Position the axes of rotation
- Activate a datum shift if required
- Define Cycle 19 **WORKING PLANE**
- Position all principal axes (X, Y, Z) in order to activate the compensation
- Define Cycle 19 with other angles, if required
- Reset Cycle 19 by programming 0° for all tilt axes
- In order to deactivate the working plane, define Cycle 19 again
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

You can define the preset in the following ways:

- Manually by touch-off
- Controlled with a HEIDENHAIN 3-D touch probe
- Automatically with a HEIDENHAIN 3-D touch probe

Further information: User's Manual for Setup, Testing and Running NC Programs
Further information: "Touch Probe Cycles: Automatic Presetting", Page 431

7.9 PRESETTING (Cycle 247, DIN/ISO: G247)

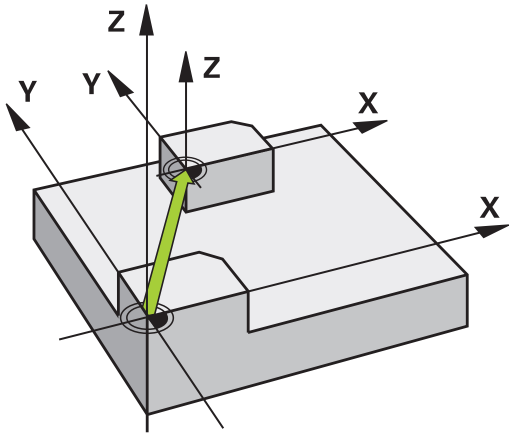
Effect

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:

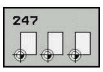
This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

When activating a preset from the preset table, the control resets the datum shift, mirroring, rotation, scaling factor and axis-specific scaling factor.

If you activate preset number 0 (line 0), then you activate the preset that you last set in the **Manual operation** or **Electronic handwheel** operating mode.

Cycle 247 is also effective in the Test Run operating mode.

Cycle parameters



- **Number for preset?**: Enter the number of the desired preset from the preset table. Alternatively, you can press the **SELECT** soft key and directly select the desired preset from the preset table. Input range: 0 to 65535

Example

```
13 CYCL DEF 247 PRESETTING
    Q339=4      ;PRESET NUMBER
```

Status displays

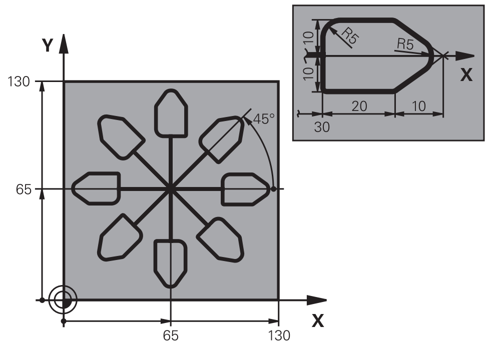
In the additional status display (**STATUS POS.**) the control shows the active preset number in addition to the **Datum** dialog.

7.10 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	

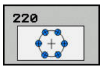
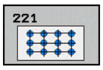
8

Fixed Cycles: Pattern definitions

8.1 Fundamentals

Overview

The control provides three cycles for machining point patterns:

Soft key	Cycle	Page
	220 POLAR PATTERN	236
	221 CARTESIAN PATTERN	239
	224 DATAMATRIX CODE PATTERN	241

You can combine Cycles 220, 221, and 224 with the following fixed cycles:

Cycle 200	**DRILLING**
Cycle 201	**REAMING**
Cycle 203	**UNIVERSAL DRILLING**
Cycle 205	**UNIVERSAL PECKING**
Cycle 208	**BORE MILLING**
Cycle 240	**CENTERING**
Cycle 251	**RECTANGULAR POCKET**
Cycle 252	**CIRCULAR POCKET**

You can combine Cycles 220 and 221 with the following fixed cycles only:

Cycle 202	**BORING**
Cycle 204	**BACK BORING**
Cycle 206	**TAPPING**
Cycle 207	**RIGID TAPPING**
Cycle 209	**TAPPING W/ CHIP BRKG**
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 MLLNG/CNTSNKG**
Cycle 264	**THREAD DRILLNG/MLLNG**
Cycle 265	**HEL. THREAD DRLG/MLG**
Cycle 267	**OUTSIDE THREAD MLLNG**

> If you have to machine irregular point patterns, use **CYCL CALL PAT** to develop point tables.
>
> More regular point patterns are available with the **PATTERN DEF** function.

Further information: "Point tables", Page 68

Further information: "Pattern definition with PATTERN DEF", Page 61

8.2 POLAR PATTERN (Cycle 220, DIN/ISO: G220, 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:

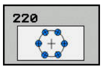
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle 220 is DEF-active. In addition, Cycle 220 automatically calls the last defined fixed cycle.

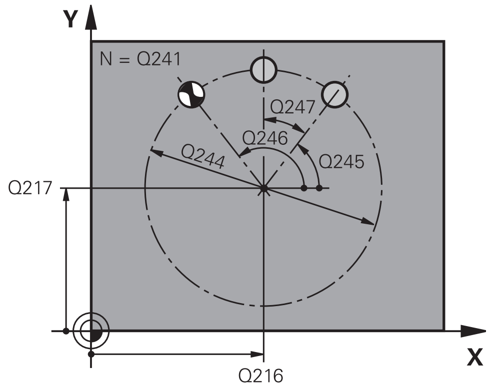
If you combine Cycle 220 or Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface and the 2nd set-up clearance that were defined in Cycle 220 or 221 will be effective. This applies within the NC program until the affected parameters are overwritten 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 calls with **CYCL CALL** and **M99** will use **Q203**=-5. Cycles 220 and 221 overwrite the above-mentioned parameters of **CALL**-active machining cycles (if the same input parameters have been programmed in both cycles).

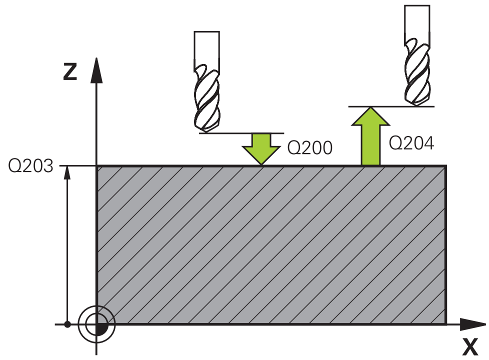
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q216 Center in 1st axis?** (absolute): Pitch circle center in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q217 Center in 2nd axis?** (absolute): Pitch circle center in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q244 Pitch circle diameter?**: Diameter of the pitch circle. Input range: 0 to 99999.9999
- **Q245 Starting angle?** (absolute): Angle between the principal 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 principal 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 an angle step of 0, the control will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the control will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). 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





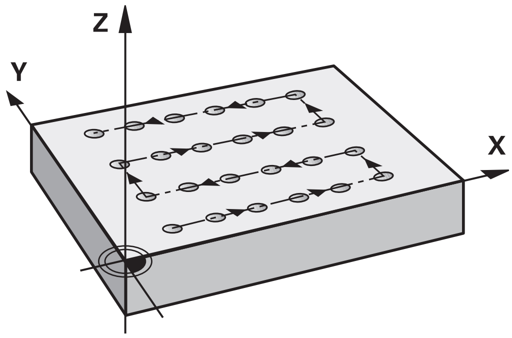
Example

```
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

8.3 LINEAR POINT PATTERN (Cycle 221, DIN/ISO: G221, 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:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

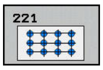
Cycle 221 is DEF-active. In addition, Cycle 221 automatically calls the last defined fixed cycle.

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.

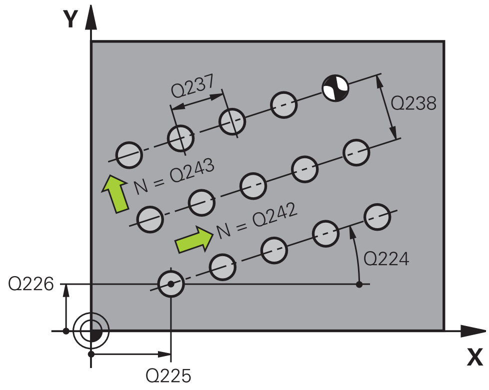
The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

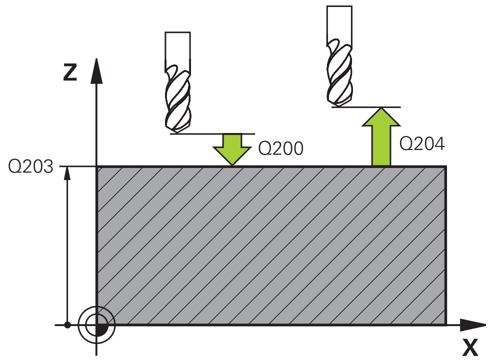
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q225 Starting point in 1st axis?** (absolute): Coordinate of the starting point in the principal axis of the working plane
- **Q226 Starting point in 2nd axis?** (absolute): Coordinate of the starting point in the secondary 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 (row)
- **Q243 Number of lines?**: Number of lines (rows)
- **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





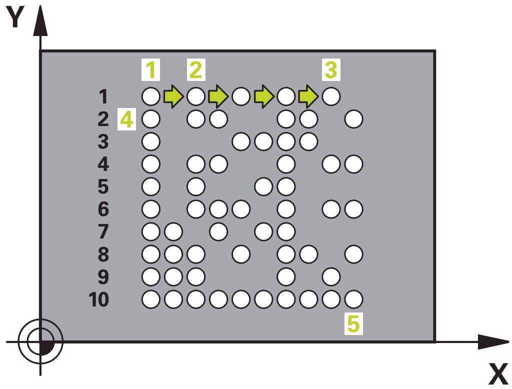
Example

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

8.4 DATAMATRIX CODE PATTERN (Cycle 224, DIN/ISO: G224, option 19)

Cycle run

With Cycle 224 **DATAMATRIX CODE PATTERN**, you can convert text to a so-called DataMatrix code. This code will be used as a point pattern for a previously defined fixed cycle.



1 The control automatically moves the tool from its current position to the programmed starting point. This point is always located in the lower left corner.

   Sequence:

   - Move to the 2nd set-up clearance (spindle axis)
   - Approach the starting point in the machining plane
   - Move to the Safety clearance above the workpiece surface (spindle axis)

2 Then, the control moves the tool in the positive direction of the secondary axis to the first starting point **1** in the first row

3 From this position, the control executes the last defined fixed machining cycle

4 Then, the control moves the tool in the positive direction of the principal axis to the second starting point **2** of the next machining operation. The tool stops at the 1st set-up clearance

5 This procedure will be repeated until all machining operations in the first row have been completed. The tool is located above the last point **3** of the first row

6 Then, the control moves the tool in the negative direction of the principal and secondary axes to the first starting point **4** of the next row

7 Then, the next points are machined

8 These steps are repeated until the entire DataMatrix code has been completed. Machining stops in the lower right corner **5**

9 Finally, the control retracts the tool to the programmed 2nd set-up clearance

Please note while programming!

NOTICE

Danger of collision!

If you combine Cycle 224 with one of the fixed cycles, the **Safety clearance**, coordinate surface and 2nd set-up clearance that you defined in Cycle 224 will be effective for the selected fixed cycle.

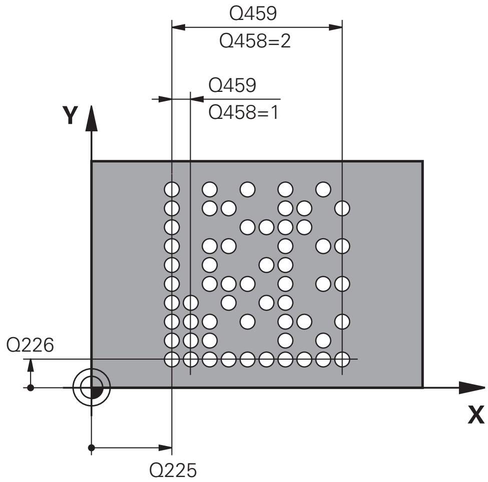
- Check the machining sequence using a graphic simulation
- Carefully test the NC program or program section in **Program run, single block** operating mode

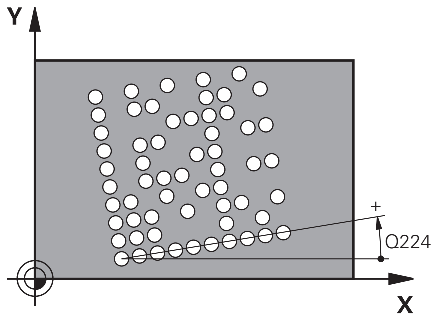
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

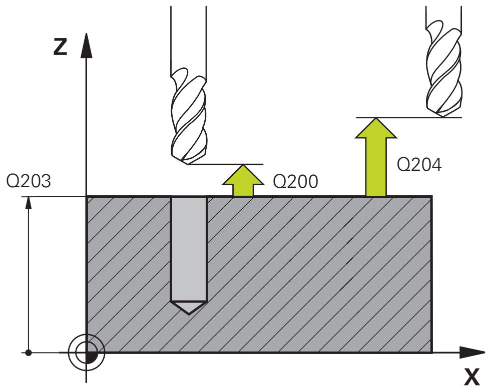
Cycle 224 is DEF-active. In addition, Cycle 224 automatically calls the last defined fixed cycle.

Cycle parameters

- **Q225 Starting point in 1st axis?** (absolute): Coordinate in the lower left corner of the code in the principal axis
- **Q226 Starting point in 2nd axis?** (absolute): Definition of a coordinate in the lower left corner of the code in the secondary axis
- **QS501 Text input?** Enter the text to be converted, in quotation marks. Maximum text length: 255 characters
- **Q458 Cell size/Pattern size(1/2)?**: Define how the DataMatrix code will be described in **Q459**:
  1: Cell spacing
  2: Pattern size
- **Q459 Size for pattern?** (incremental): Defines the cell spacing or the pattern size:
  If **Q458=1**: Spacing between the first and second cell (based on the cell centers)
  If **Q458=2**: Spacing between the first and last cell (based on the cell centers)
  Input range: 0 to 99999.9999
- **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





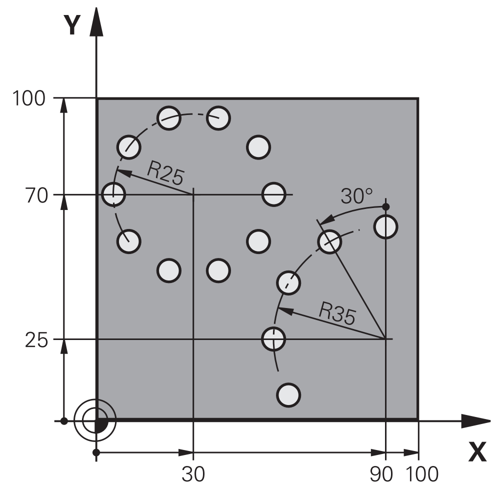


Example

```
54 CYCL DEF 224  DATAMATRIX CODE
   PATTERN
   Q225=+0     ;STARTNG PNT 1ST AXIS
   Q226=+0     ;STARTNG PNT 2ND AXIS
   QS501="ABC";TEXT
   Q458=+1     ;SIZE SELECTION
   Q459=+1     ;SIZE
   Q224=+0     ;ANGLE OF ROTATION
   Q200=+2     ;SET-UP CLEARANCE
   Q203=+0     ;SURFACE COORDINATE
   Q204=50     ;2ND SET-UP CLEARANCE
```

8.5 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	

9

Fixed Cycles: Contour Pocket

9.1 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.

> 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 permitted—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
...
```

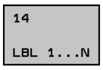
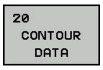
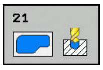
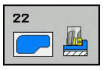
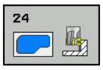
```
60 LBL 0
...
99 END PGM SL2 MM
```

Characteristics of the fixed cycles

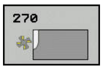
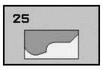
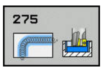
- The control automatically positions the tool to the set-up clearance before each cycle. You must move the tool to a safe position before the cycle call
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them
- The radius of inside 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.

Overview

Soft key	Cycle	Page
	14 CONTOUR (compulsory)	251
	20 CONTOUR DATA (compulsory)	256
	21 PILOT DRILLING (optional)	258
	22 ROUGH-OUT (compulsory)	260
	23 FLOOR FINISHING (optional)	265
	24 SIDE FINISHING (optional)	267

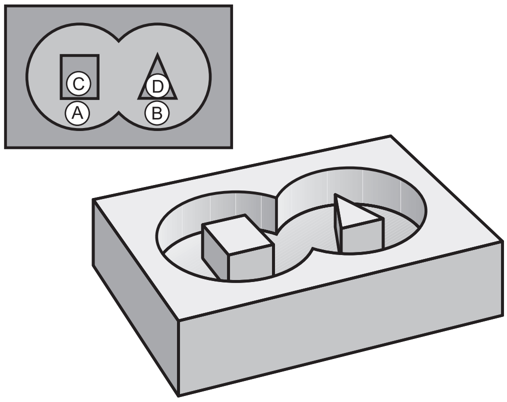
Enhanced cycles:

Soft key	Cycle	Page
	270 CONTOUR TRAIN DATA	270
	25 CONTOUR TRAIN	272
	275 TROCHOIDAL SLOT	276
	276 THREE-D CONT. TRAIN	282

9.2 CONTOUR (Cycle 14, DIN/ISO: G37)

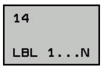
Please note while programming:

All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR.



> This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
>
> Cycle 14 is DEF-active which means that it becomes effective as soon as it is defined in the NC program.
>
> You can list up to 12 subprograms (subcontours) in Cycle 14.

Cycle parameters

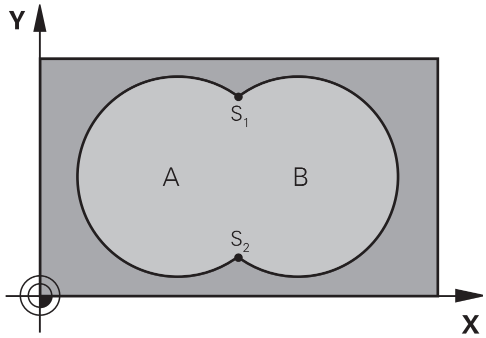


- **Label numbers for the contour**: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm each input with the ENT key. Confirm each input with the **END** key. Input of up to 12 subprogram numbers: 1 to 65 535

9.3 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 GEOMETRY
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

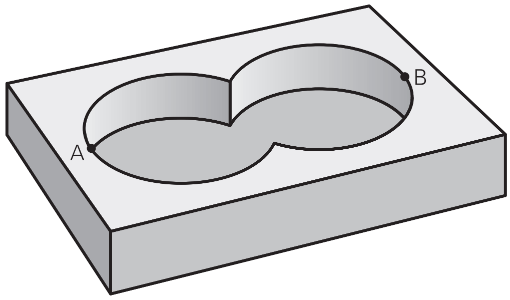
Subprogram 2: Pocket B

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

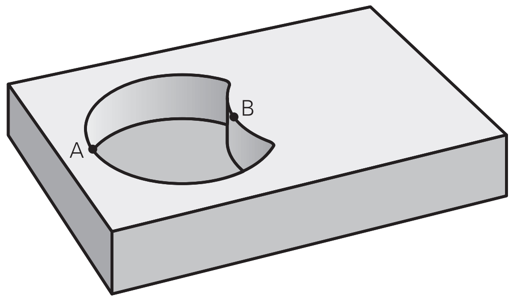
Surface B:

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

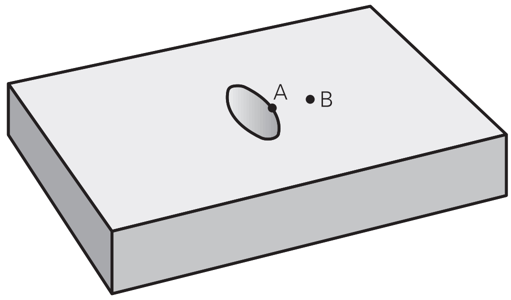
Surface B:

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

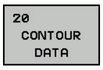
9.4 CONTOUR DATA (Cycle 20, DIN/ISO: G120, option 19)

Please note while programming:

Use Cycle 20 to program machining data for the subprograms describing the subcontours.

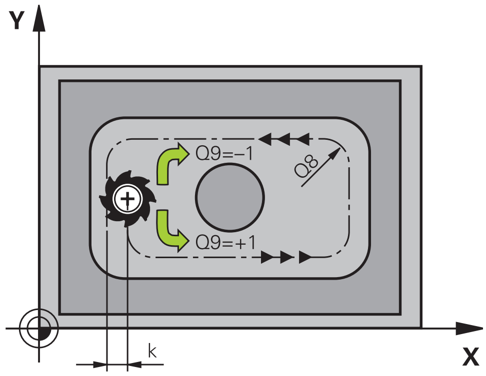
> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
>
> Cycle 20 is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
>
> The machining data entered in Cycle 20 are valid for Cycles 21 to 24.
>
> The algebraic sign for the DEPTH cycle 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.

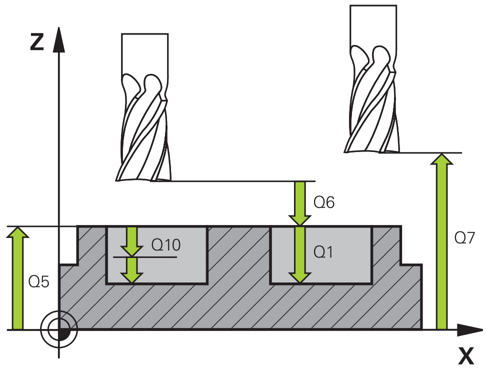
Cycle parameters



- **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

9.5 PILOT DRILLING (Cycle 21, DIN/ISO: G121, 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:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

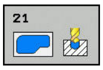
When calculating the infeed points, the control does not account for the delta value **DR** programmed in a **TOOL CALL** block.

In narrow areas, the control may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

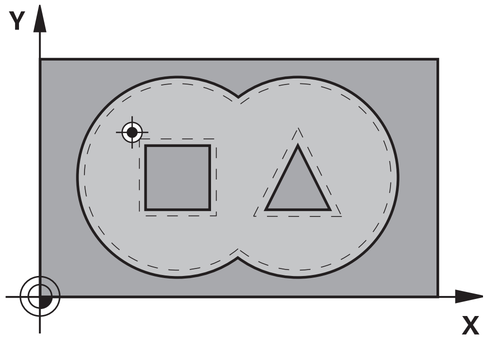
If **Q13**=0, the control uses the data of the tool that is currently in the spindle.

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.

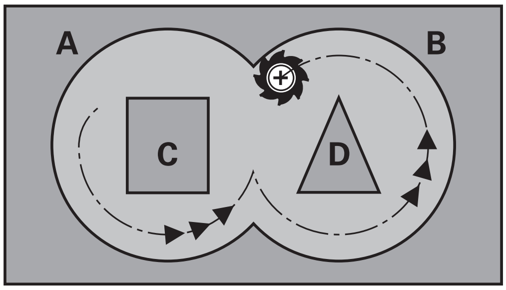


Example

58 CYCL DEF 21 PILOT DRILLING	
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q13=1	;ROUGH-OUT TOOL

9.6 ROUGHING (Cycle 22, DIN/ISO: G122, 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 After reaching the first plunging depth, the tool mills the contour in an outward direction at the programmed milling feed rate **Q12**
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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.

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.

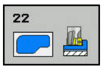
During fine roughing, the control does not take a defined wear value **DR** of the coarse roughing tool into account.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.

You define the plunging behavior of Cycle 22 with parameter **Q19** and in the **ANGLE** and **LCUTS** columns of the tool table:

- If **Q19**=0 is defined, the tool will always plunge perpendicularly, even if a plunge angle (**ANGLE**) was defined for the active tool
- If you define **ANGLE**=90°, the control will plunge perpendicularly. The reciprocation feed rate **Q19** is used as plunging feed rate
- If the reciprocation feed rate **Q19** is defined in Cycle 22 and **ANGLE** is between 0.1 and 89.999 in the tool table, the tool plunges helically using the defined **ANGLE**
- If the reciprocation feed is defined in Cycle 22 and no **ANGLE** 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**))

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

9.7 FLOOR FINISHING (Cycle 23, DIN/ISO: G123, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

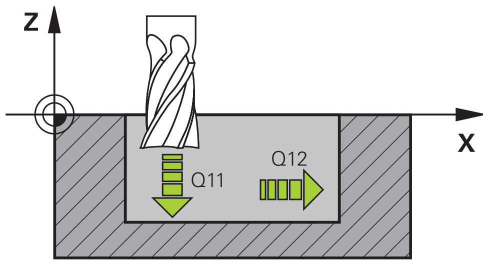
The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

The approaching radius for pre-positioning to the final depth is permanently defined and independent of the plunging angle of the tool.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.

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. ROUGHING
Q208=9999	;RETRACTION FEED RATE

9.8 SIDE FINISHING (Cycle 24, DIN/ISO: G124, 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The sum of finishing allowance for the side (**Q14**) and the radius of the finish mill must be smaller than the sum of allowance for side (**Q3**, Cycle 20) and the radius of the rough mill.

If no allowance has been defined in Cycle 20, the control issues the error message "Tool radius too large".

The finishing allowance for the 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:

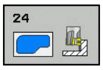
- define the contour to be milled as a single island (without pocket limit)
- enter the finishing allowance (**Q3**) in Cycle 20 to be greater than the sum of the finishing allowance **Q14** + radius of the tool being used

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.

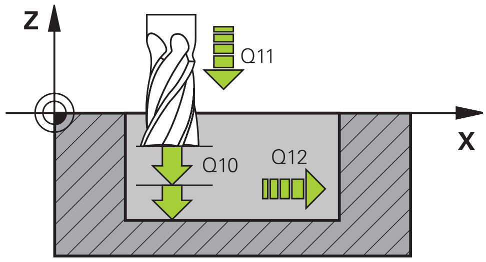
The starting point calculated by the control also depends on the machining sequence. If you select the finishing cycle with the GOTO key and then start the NC program, the starting point can be at a different location from where it would be if you execute the NC program in the defined sequence.

If **M110** is activated during operation, the feed rate of compensated circular arcs within will be reduced accordingly.

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 finishing allowance for the 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 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 the number of a tool with the radius 0. This is usually the tool numbered 0.



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?

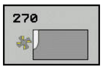
9.9 CONTOUR TRAIN DATA (Cycle 270, DIN/ISO: G270, option 19)

Please note while programming:

You can use this cycle to specify various properties of Cycle 25 CONTOUR TRAIN.

> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
>
> Cycle 270 is DEF-active, which means that it becomes effective as soon as it is defined in the NC program.
>
> If Cycle 270 is used, do not define any radius compensation in the contour subprogram.
>
> Define Cycle 270 before Cycle 25.

Cycle parameters



- **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

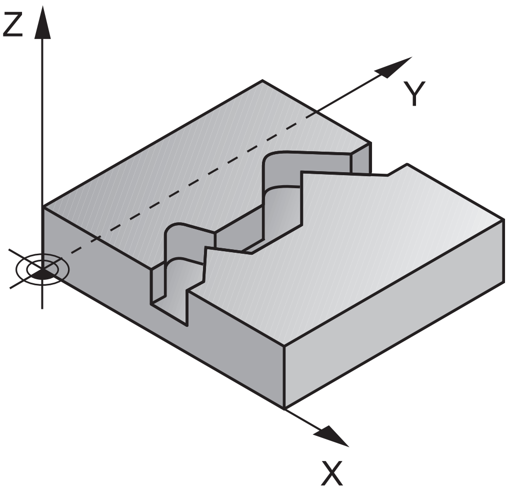
Example

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

9.10 CONTOUR TRAIN (Cycle 25, DIN/ISO: G125, 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 (run a graphic simulation of the contour before execution)
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked
- Machining can done throughout by up-cut or by climb milling. The type of milling will even be retained if the contours were mirrored
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.

Please note while programming!

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

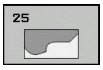
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

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.

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 beginning and end of the end of a contour area. The control always extends the tool path parallel to the contour. Input range: 0 to 99.999

9.11 TROCHOIDAL SLOT (Cycle 275, DIN/ISO: G275, 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 the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached.

Finishing 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 GEOMETRY
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 the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached.

Finishing with open slots

5 If a finishing allowance has been defined, the control finishes the slot walls (in multiple infeeds if specified). The control approaches the slot wall starting from the defined starting point of the **APPR** block. Climb or up-cut milling is taken into consideration

Please note while programming:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

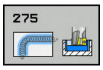
Define the center line of the slot with all available path functions in the contour subprogram.

The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.

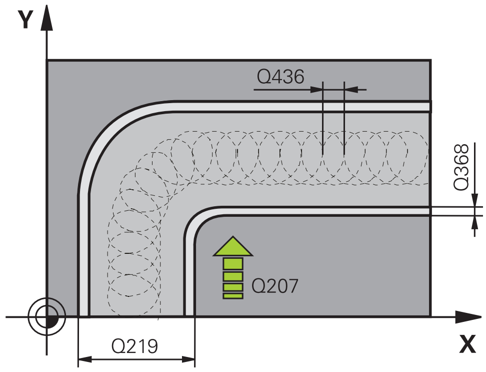
In conjunction with Cycle 275, the control does not require Cycle 20 CONTOUR DATA.

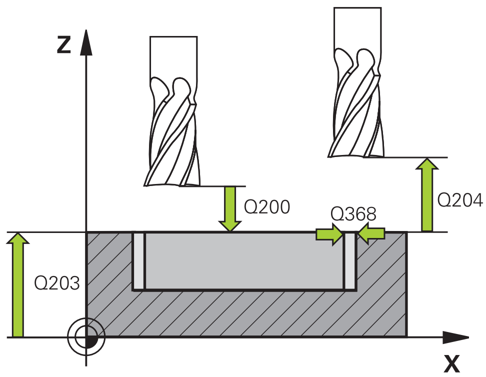
The starting point of a closed slot must not be located in a contour corner.

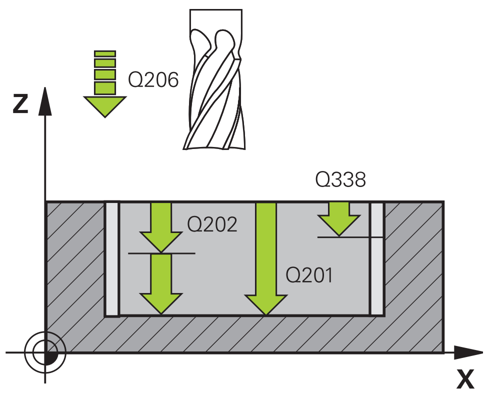
Cycle parameters



- **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 secondary 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**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  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 with a single 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**

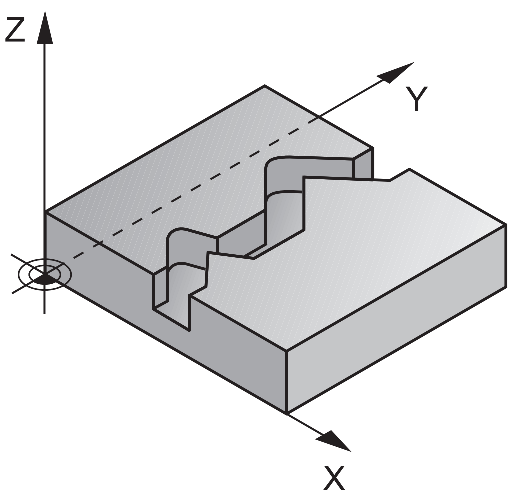
Example

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 MILLING
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

9.12 THREE-D CONT. TRAIN (Cycle 276, DIN/ISO: G276, 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 Type of approach. The control then moves the tool to the first plunging depth
2. According to the previously defined Cycle 270 **CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3. At the end of the contour, the tool will be retracted as defined in Cycle 270 **CONTOUR TRAIN DATA**
4. Finally, the control retracts the tool to the clearance height.

Machining a contour with infeed: Milling depth **Q1** not equal to 0 and plunging depth **Q10** are defined

1. The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle 270 **CONTOUR TRAIN DATA**, e.g. in Type of approach. The control then moves the tool to the first plunging depth
2. According to the previously defined Cycle 270 **CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3. If you selected machining with Climb milling and up-cut milling (**Q15**=0), the control will perform a reciprocation movement. The infeed movement (plunging) will be performed at the end and at the starting point of the contour. If **Q15** is not equal to 0, the tool is moved to clearance height and 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:

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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 you are 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.

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

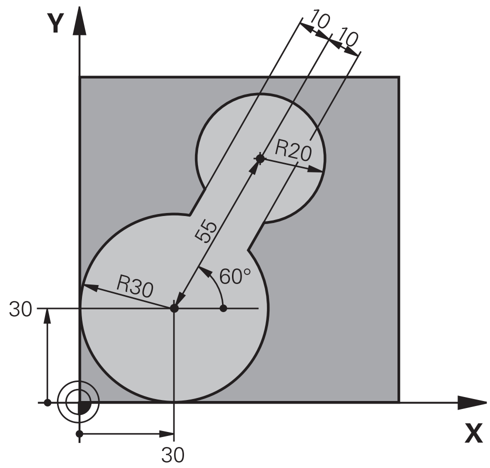
Example

```
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 beginning and end of the end of a contour area. The control always extends the tool path parallel to the contour. Input range: 0 to 99.999

9.13 Programming Examples

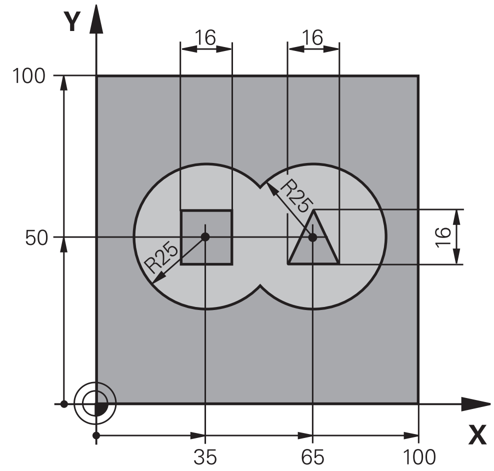
Example: Roughing-out and fine-roughing a pocket



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 GEOMETRY	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	

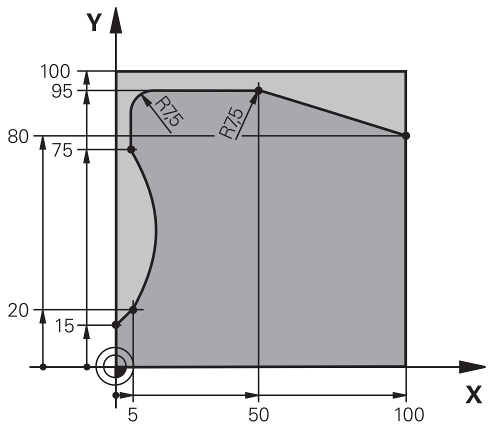
Example: Pilot drilling, roughing-out and finishing overlapping contours



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 GEOMETRY	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 GEOMETRY	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	

10

Machining Cycles: Optimized contour milling

10.1 OCM cycles (option 167)

OCM fundamentals

General information

Using OCM cycles (**Optimized Contour Milling**), you can combine subcontours to form complex contours. These cycles are more powerful than Cycles 22 to 24. OCM cycles provide the following additional functions:

- When roughing, the control will maintain the specified tool angle precisely
- Besides pockets, you can also machine islands and open pockets

> You can program up to 16 384 contour elements in one OCM cycle.
>
> OCM cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, 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.

Contact angle

When roughing, the control will retain the tool angle precisely. The tool angle can be defined implicitly by specifying an overlap factor. The maximum overlap factor is 1; this corresponds to an angle of 90° max.

Contour

Specify the contour with **CONTOUR DEF**. The first contour may be a pocket or a boundary. The next contours can be programmed as islands or pockets.

To program open pockets, use a boundary and an island.

Proceed as follows:

- Program **CONTOUR DEF**
- Define the first contour as a pocket and the second one as an island
- Define the **OCM CONTOUR DATA** cycle
- In cycle parameter **Q569**, program the value 1
- > The control will interpret the first contour as an open boundary instead of a pocket. Thus, the open boundary and the island programmed subsequently are combined to form an open pocket.

An example is provided after the description of the OCM cycles, see "Example: Open pocket and fine roughing with OCM cycles", Page 306

> Subsequently defined contours that are outside the first contour will not be considered.

Closed pockets can also be defined in Cycle 14.

The machining dimensions, such as milling depth, finishing allowances and clearance height, can be entered in Cycle 271 as **OCM CONTOUR DATA**.

Machining operation

When roughing, these cycles allow you to use larger tools for the first roughing passes and then smaller tools to take off the residual material. For finishing, the material roughed out will be taken into consideration.

Example

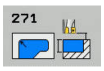
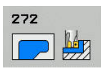
You defined a Ø20 mm rough-out tool. For roughing, this results in minimum inside radii of 10 mm (cycle parameter **Q578** Radius factor on inside corners will not be taken into account in this example). In the next step, you will finish the contour. For this purpose, define a Ø10 mm finishing cutter. In this case, the maximum inside radii would be 5 mm. Finishing cycles will also consider the previous machining steps, depending on **Q438**, so that the smallest finishing inside radii will be 10 mm. Thus, the finishing cutter will be protected from overload.

Program structure: Machining with OCM cycles

```
0 BEGIN PGM  SL2 MM
...
12 CONTOUR DEF ...
13 CYCL DEF 271 OCM CONTOUR DATA ...
...
16 CYCL DEF 272  OCM ROUGHING ...
17 CYCL CALL
...
18 CYCL DEF 273 OCM FINISHING FLOOR  ...
19 CYCL CALL
...
22 CYCL DEF 274  OCM FINISHING SIDE ...
23 CYCL CALL
50 L  Z+250 R0  FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM  SL2 MM
```

Overview

OCM cycles:

Soft key	Cycle	Page
	271 OCM CONTOUR DATA	297
	272 OCM ROUGHING	299
	273 OCM FINISHING FLOOR	302
	274 OCM FINISHING SIDE	304

10.2 OCM CONTOUR DATA (Cycle 271, DIN/ISO: G271, option 167)

Cycle run

Use Cycle 271 **OCM CONTOUR DATA** to program machining data for the contour or the subprograms describing the subcontours. In addition, Cycle 271 allows you to define an open boundary for a pocket.

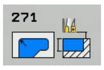
Please note while programming!

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

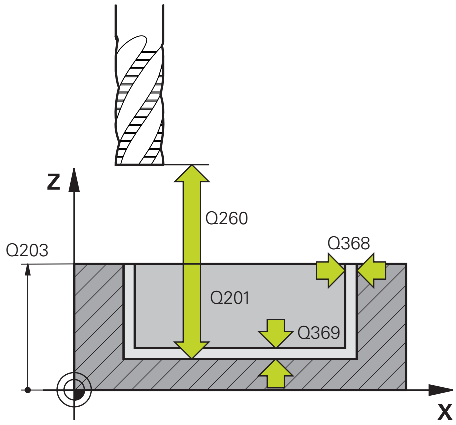
Cycle 271 is DEF-active, which means that it becomes active as soon as it is defined in the NC program.

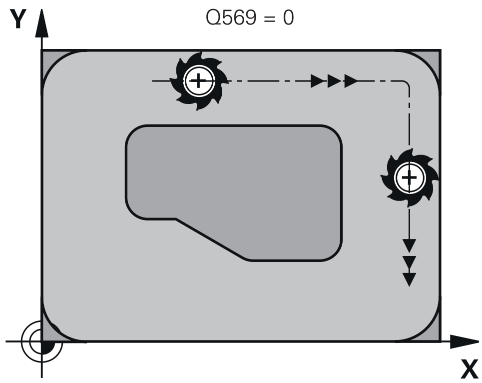
The machining data entered in Cycle 271 are valid for Cycles 272 to 274.

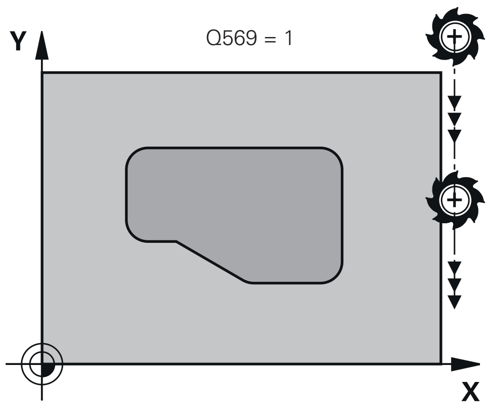
Cycle parameters



- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: –99999.9999 to 0
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q260 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
- **Q578 Radius factor on inside corners?** The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**. Input range: 0.05 to 0.99
- **Q569 Is the first pocket a boundary?** Define the boundary:
  0: The first contour in CONTOUR DEF is interpreted as a pocket.
  1: The first contour in CONTOUR DEF is interpreted as an open boundary.







Example

59 CYCL DEF 271 OCM CONTOUR DATA
Q203=+0 ;SURFACE COORDINATE
Q201=-20 ;DEPTH
Q368=+0 ;ALLOWANCE FOR SIDE
Q369=+0 ;ALLOWANCE FOR FLOOR
Q260=+100 ;CLEARANCE HEIGHT
Q578=+0.2 ;INSIDE CORNER FACTOR
Q569=+0 ;OPEN BOUNDARY

10.3 OCM ROUGHING (Cycle 272, DIN/ISO: G272, option 167)

Cycle run

Use Cycle 272 **OCM ROUGHING** to define the technology data for roughing.

Before calling Cycle 272 you need to program further cycles:

- **CONTOUR DEF**, alternatively Cycle 14 **CONTOUR GEOMETRY**
- Cycle 271 **OCM CONTOUR DATA**

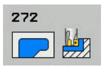
1. The tool uses positioning logic to move to the starting point.
2. The control determines the starting point automatically based on the pre-positioning and the programmed contour.
   - If you program **Q569**=0, the tool plunges into the material in a helical movement to reach the first plunging depth. The finishing allowance for the side is taken into account
   - If you program **Q569**=1, the tool plunges outside the open boundary
3. After reaching the first plunging depth, the tool mills the contour in an outward or inward direction (depending on **Q569**) at the programmed milling feed rate **Q207**
4. In the next step, the control moves the tool to the next plunging depth and repeats the roughing procedure until the programmed depth is reached
5. Finally, the tool retracts in the tool axis to the clearance height.

Please note while programming!

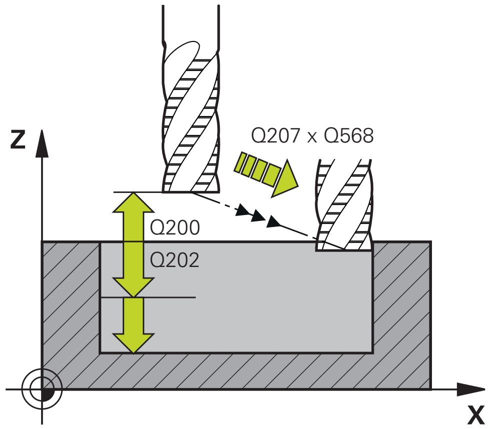
> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
>
> CONTOUR DEF will reset the tool radius that was used last. If you run this machining cycle with Q438=–1 after CONTOUR DEF, the control assumes that no pre-machining has taken place yet.
>
> If required, use a center-cut end mill (ISO 1641).
>
> If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.

> Define the plunging behavior of Cycle 272 in the **ANGLE** and **LCUTS** columns of the tool table:
>
> - If **ANGLE** is between 0.1° and 89.999° in the tool table, the control plunges helically using the defined **ANGLE**
> - If **ANGLE** is less than 0.1° or larger than 90° in the tool table, the control displays an error message
> - If the geometry of the workpiece (slot) prevents helical plunging, the control displays a note that plunging is not possible at this position. You can then use a smaller tool for re-machining.

Cycle parameters



- **Q202 Plunging depth?** (incremental): Infeed per cut. Input range: 0 to 99999.9999
- **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: 0.01 to 1; alternatively **PREDEF**
- **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**
- **Q568 Factor for plunging feed rate?** Factor by which the control reduces the feed rate **Q207** for downfeed into the material. Input range: 0.1 to 1
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. In mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q438 Number/name of 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 in Cycle 272 was the rough-out tool (default behavior)
  Q438=0: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0.



Example

59 CYCL DEF 272 OCM ROUGHING	
Q202=+5	;PLUNGING DEPTH
Q370=+0.4	;TOOL PATH OVERLAP
Q207=+500	;FEED RATE MILLING
Q568=+0.6	;PLUNGING FACTOR
Q253=+750	;F PRE-POSITIONING
Q200=+2	;SAFETY CLEARANCE
Q438=-1	;ROUGH-OUT TOOL
Q577=+0.2	;APPROACH RADIUS FACTOR
Q351=+1	;CLIMB OR UP-CUT

- **Q577 Factor for appr./dept. radius?** Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input range: 0.15 to 0.99
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)

10.4 OCM FINISHING FLOOR (Cycle 273, DIN/ISO: G273, option 167)

Cycle run

With Cycle 273 **OCM FINISHING FLOOR**, you can clear the finishing allowance for side that is programmed in Cycle 271.

Before calling Cycle 273 you need to program further cycles:

- **CONTOUR DEF**, alternatively Cycle 14 **CONTOUR GEOMETRY**
- Cycle 271 **OCM CONTOUR DATA**
- Cycle 272 **OCM ROUGHING**, if applicable**OCM ROUGHING**

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 **Q385**
3 The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically
4 The tool clears the finishing allowance remaining from rough-out
5 Finally, the tool retracts in the tool axis to the clearance height

Please note while programming!

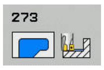
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour.

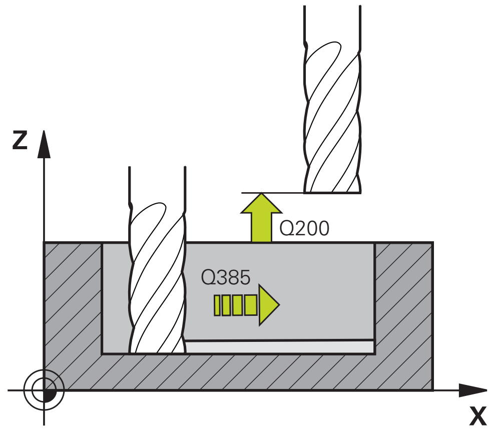
For finishing with Cycle 273, the tool always works in climb milling mode.

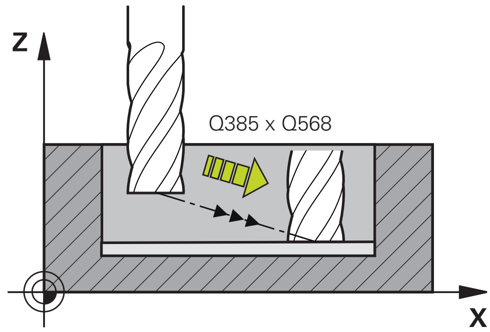
Specify a rough-out tool in cycle parameter **Q438**, otherwise, the control will generate an error message.

Cycle parameters



- **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: 0.0001 to 1.9999; alternatively **PREDEF**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during floor finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q568 Factor for plunging feed rate?** Factor by which the control reduces the feed rate **Q385** for downfeed into the material. Input range: 0.1 to 1
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. In mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q438 Number/name of 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)





Example

60 CYCL DEF 273 OCM FINISHING FLOOR	
Q370=+1	;TOOL PATH OVERLAP
Q385=+500	;FINISHING FEED RATE
Q568=+0.3	;PLUNGING FACTOR
Q253=+750	;F PRE-POSITIONING
Q200=+2	;SET-UP CLEARANCE
Q438=-1	;ROUGH-OUT TOOL

10.5 OCM FINISHING SIDE (Cycle 274, DIN/ISO: G274, option 167)

Cycle run

With Cycle 274 **OCM FINISHING SIDE**, you can clear the finishing allowance for the side that is programmed in Cycle 271. You can run this cycle in climb or up-cut milling.

Before calling Cycle 274 you need to program further cycles:

- **CONTOUR DEF**, alternatively Cycle 14 **CONTOUR GEOMETRY**
- Cycle 271 **OCM CONTOUR DATA**
- Cycle 272 **OCM ROUGHING**, if applicable
- Cycle 273 **OCM FINISHING FLOOR**, if applicable

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 tool approaches and moves along the contour helically on a tangential arc until the entire contour is finished. Each subcontour is finished separately
4. Finally, the tool retracts in the tool axis to the clearance height.

Cycle 274 can also be used for contour milling.

Proceed as follows:

- ▶ Define the contour to be milled as a single island (without pocket boundary)
- ▶ Enter the finishing allowance (**Q368**) in Cycle 271 to be greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Please note while programming!

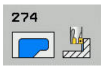
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The finishing allowance for the side **Q14** is left over after finishing. It must be smaller than the allowance in Cycle 271.

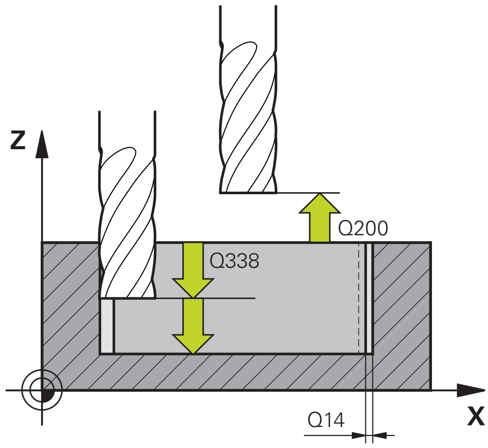
The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour and the allowance programmed in Cycle 271.

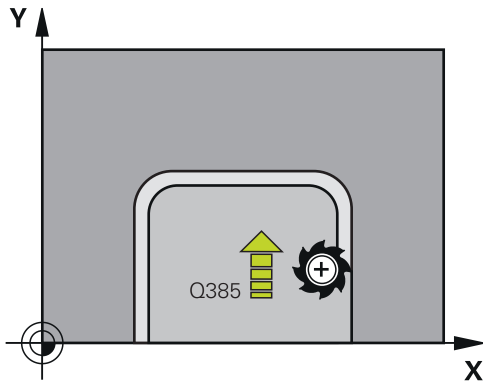
Specify a rough-out tool in cycle parameter **Q438**, otherwise, the control will generate an error message.

Cycle parameters



- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. **Q338**=0: Finishing with a single infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side finishing. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool when approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. In mm/min. Input range: 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q14 Finishing allowance for side?** (incremental): The finishing allowance for the side **Q14** is left over after finishing. (This allowance must be smaller than the allowance in Cycle 271.) Input range: –99999.9999 to 99999.9999
- **Q438 Number/name of 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)
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation. The spindle rotation is taken into account:
  +1 = Climb milling
  -1 = Up-cut milling
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)





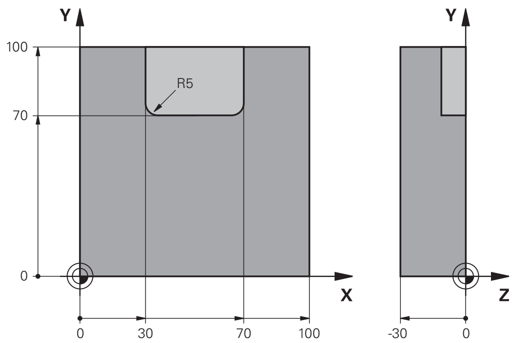
Example

61 CYCL DEF 274 OCM FINISHING SIDE	
Q338=+0	;INFEED FOR FINISHING
Q385=+500	;FINISHING FEED RATE
Q253=+750	;F PRE-POSITIONING
Q200=+2	;SET-UP CLEARANCE
Q14=+0	;ALLOWANCE FOR SIDE
Q438=-1	;NUMBER/NAME OF ROUGH-OUT TOOL?
Q351=+1	;CLIMB OR UP-CUT

10.6 Programming examples

Example: Open pocket and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will program an open pocket. For this purpose, use a boundary and an island.



Program sequence

- Tool call: roughing cutter
- Program **CONTOUR DEF**
- Define Cycle 271
- Define and call Cycle 272
- Tool call: finishing cutter
- Define and call Cycle 273
- Define and call Cycle 274

0 BEGIN PGM OCM_POCKET MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL "MILL_D20" Z S8000 F1500	Tool call, diameter 20
4 M3	
5 L Z+250 R0 FMAX	
6 L X+0 Y+0 R0 FMAX	
7 CONTOUR DEF P1 = LBL 1 I2 = LBL 2	
8 CYCL DEF 271 OCM CONTOUR DATA	Define machining parameters
Q203=+0 ;SURFACE COORDINATE	
Q201=-10 ;DEPTH	
Q368=+0.5 ;ALLOWANCE FOR SIDE	
Q369=+0.5 ;ALLOWANCE FOR FLOOR	
Q260=+100 ;SICHERE HOEHE	
Q578=+0.2 ;INSIDE CORNER FACTOR	
Q569=+1 ;OPEN BOUNDARY	
9 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+5 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207= AUTO ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=+0 ;ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
10 CYCL CALL	Cycle call
11 TOOL CALL "MILL_D8" Z S8000 F1500	Tool call, diameter 8

12 M3	
13 L Z+250 R0 FMAX	
14 L X+0 Y+0 R0 FMAX	
15 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+5 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207= AUTO ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
QS438="MILL_D20" ;ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
16 CYCL CALL	Cycle call
17 TOOL CALL "MILL_D6_FINISH" Z S10000 F2000	Tool call, diameter 6
18 M3	
19 L Z+250 R0 FMAX	
20 L X+0 Y+0 R0 FMAX	
21 CYCL DEF 273 OCM FINISHING FLOOR	Define the floor finishing cycle
Q370=+0.8 ;TOOL PATH OVERLAP	
Q385= AUTO ;FINISHING FEED RATE	
Q568=+0.3 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=-1 ;ROUGH-OUT TOOL	
22 CYCL CALL	Cycle call
23 CYCL DEF 274 OCM FINISHING SIDE	Define the side finishing cycle
Q338=+0 ;INFEED FOR FINISHING	
Q385= AUTO ;FINISHING FEED RATE	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q14=+0 ;ALLOWANCE FOR SIDE	
QS438=-1 ;ROUGH-OUT TOOL	
Q351=+1 ;CLIMB OR UP-CUT	
24 CYCL CALL	Cycle call
25 M30	End of program
26 LBL 1	Contour subprogram 1
27 L X+0 Y+0	
28 L X+100	
29 L Y+100	
30 L X+0	
31 L Y+0	
32 LBL 0	

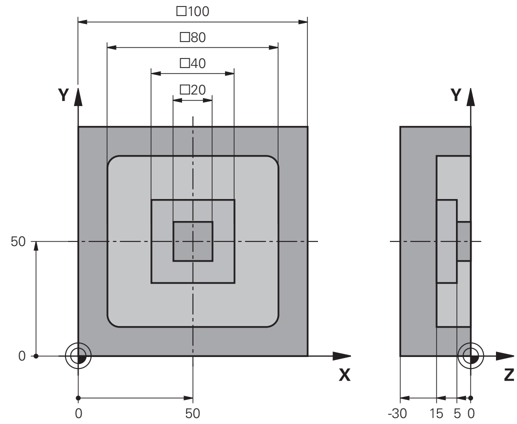
33 LBL 2	Contour subprogram 2
34 L X+0 Y+0	
35 L X+100	
36 L Y+100	
37 L X+70	
38 L Y+70	
39 RND R5	
40 L X+30	
41 L Y+100	
42 RND R5	
43 L X+0	
44 L Y+0	
45 LBL 0	
46 END PGM OCM_POCKET MM	

Example: Program various depths with OCM cycles

The following NC program illustrates the use of OCM cycles.
Define a pocket and two islands at different heights.

Program sequence

- Tool call: roughing cutter
- Program **CONTOUR DEF**
- Define Cycle 271
- Define and call Cycle 272
- Tool call: finishing cutter
- Define and call Cycle 273
- Define and call Cycle 274



0 BEGIN PGM OCM_DEPTH MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-30	Workpiece blank definition
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL "MILL_D10" Z S8000 F1500	Tool call, diameter D10
4 L Z+250 R0 FMAX M3	
5 L X+0 Y+0 R0 FMAX	
6 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 I3 = LBL 3 DEPTH5	
7 CYCL DEF 271 OCM CONTOUR DATA	Define machining parameters
Q203=+0 ;SURFACE COORDINATE	
Q201=-15 ;DEPTH	
Q368=+0.5 ;ALLOWANCE FOR SIDE	
Q369=+0.5 ;ALLOWANCE FOR FLOOR	
Q260=+100 ;SICHERE HOEHE	
Q578=+0.2 ;INSIDE CORNER FACTOR	
Q569=+0 ;OPEN BOUNDARY	
8 CYCL DEF 272 OCM ROUGHING	Define the roughing cycle
Q202=+5 ;PLUNGING DEPTH	
Q370=+0.4 ;TOOL PATH OVERLAP	
Q207= AUTO ;FEED RATE MILLING	
Q568=+0.6 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=+0 ;ROUGH-OUT TOOL	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
Q351=+1 ;CLIMB OR UP-CUT	
9 CYCL CALL	Cycle call
10 TOOL CALL "MILL_D6_FINISH" Z S10000 F2000	Tool call, diameter D6
11 M3	
12 L Z+250 R0 FMAX	
13 L X+0 Y+0 R0 FMAX	

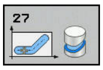
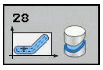
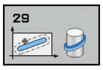
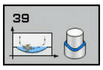
14 CYCL DEF 273 OCM FINISHING FLOOR	Define the floor finishing cycle
Q370=+0.8 ;TOOL PATH OVERLAP	
Q385= AUTO ;FINISHING FEED RATE	
Q568=+0.3 ;PLUNGING FACTOR	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q438=-1 ;ROUGH-OUT TOOL	
15 CYCL CALL	Cycle call
16 CYCL DEF 274 OCM FINISHING SIDE	Define the side finishing cycle
Q338=+0 ;INFEED FOR FINISHING	
Q385= AUTO ;FINISHING FEED RATE	
Q253=+750 ;F PRE-POSITIONING	
Q200=+2 ;SET-UP CLEARANCE	
Q14=+0 ;ALLOWANCE FOR SIDE	
QS438="MILL_D10" ;ROUGH-OUT TOOL	
Q351=+1 ;CLIMB OR UP-CUT	
17 CYCL CALL	Cycle call
18 M30	End of program
19 LBL 1	Contour subprogram 1
20 L X-40 Y-40	
21 L X+40	
22 L Y+40	
23 L X-40	
24 L Y-40	
25 LBL 0	
26 LBL 2	Contour subprogram 2
27 L X-10 Y-10	
28 L X+10	
29 L Y+10	
30 L X-10	
31 L Y-10	
32 LBL 0	
33 LBL 3	Contour subprogram 3
34 L X-20 Y-20	
35 L Y+20	
36 L X+20	
37 L Y-20	
38 L X-20	
39 LBL 0	
40 END PGM OCM_DEPTH MM	

11

Fixed Cycles: Cylindrical Surface

11.1 Fundamentals

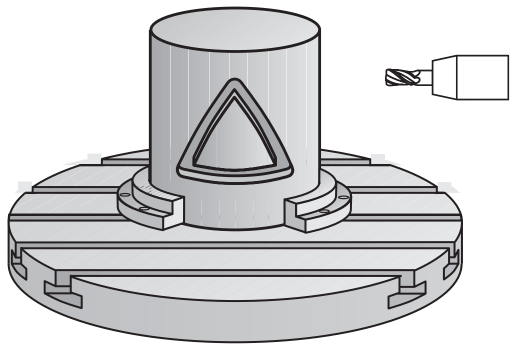
Overview of cylindrical surface cycles

Soft key	Cycle	Page
	27 CYLINDER SURFACE	313
	28 CYLINDER SURFACE Slot milling	316
	29 CYLINDER SURFACE Ridge milling	320
	39 CYLINDER SURFACE Contour	323

11.2 CYLINDER SURFACE (Cycle 27, DIN/ISO: G127, option 1)

Cycle run

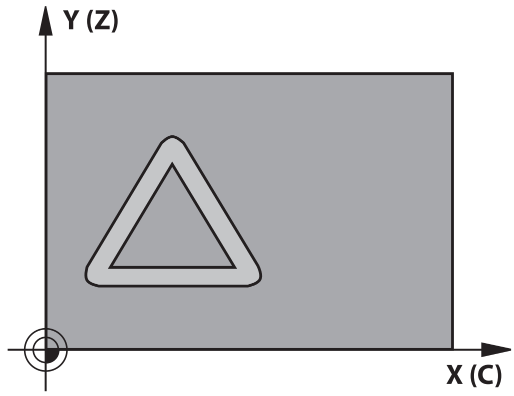
Refer to your machine manual!
The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.



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:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

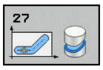
This cycle can also be used in a tilted working plane.

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 unrolled 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

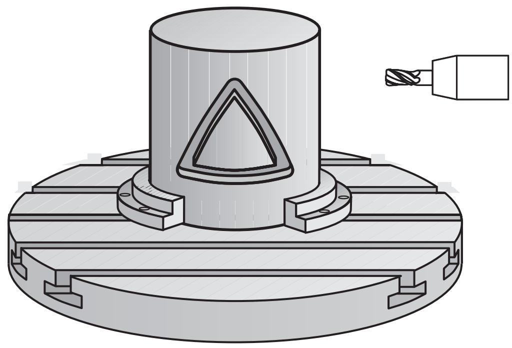
Example

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

11.3 CYLINDER SURFACE slot milling (Cycle 28, DIN/ISO: G128, option 1)

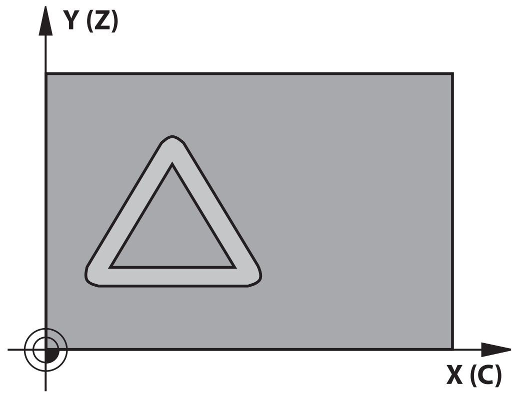
Cycle run

> Refer to your machine manual!
> The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.



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 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:

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.

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

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

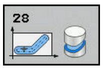
The machining time can increase if the contour consists of many non-tangential contour elements.

If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Define the approach in **apprDepCylWall** (no. 201004)

- CircleTangential:
  Tangential approach and departure
- LineNormal: The tools approaches the contour starting point on a straight line

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).

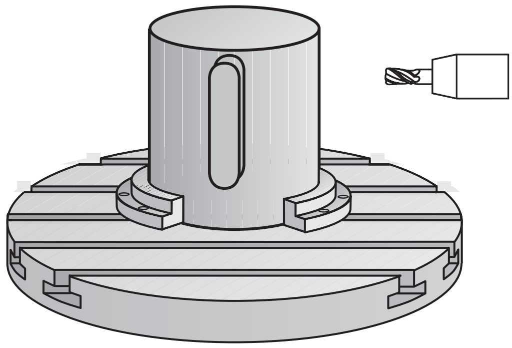
Example

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

11.4 CYLINDRICAL SURFACE ridge milling (Cycle 29, DIN/ISO: G129, option 1)

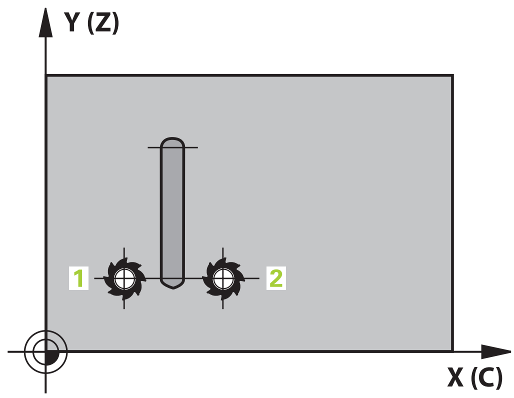
Cycle run

> Refer to your machine manual!
> The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.



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:

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.

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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).

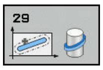
The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

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.

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

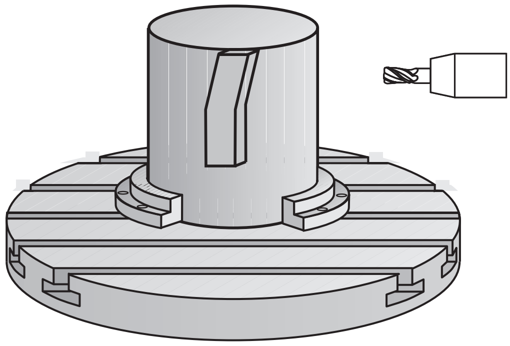
Example

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

11.5 CYLINDRICAL SURFACE CONTOUR (Cycle 39, DIN/ISO: G139, option 1)

Cycle run

Refer to your machine manual!
The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.



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:

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.

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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.

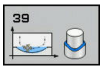
The machining time can increase if the contour consists of many non-tangential contour elements.

If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Define the approach in **apprDepCylWall** (no. 201004)

- CircleTangential:
  Tangential approach and departure
- LineNormal: The tools approaches the contour starting point on a straight line

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 unrolled 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

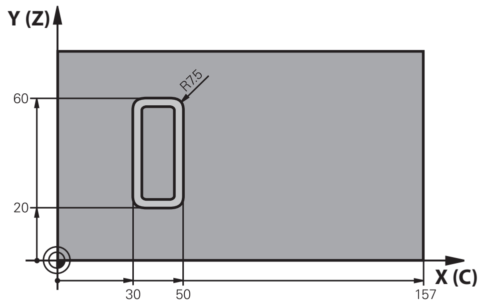
Example

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

11.6 Programming Examples

Example: Cylinder surface with Cycle 27

- Machine with B head and C table
- Cylinder centered on rotary table
- Preset is on the underside, in the center of the rotary table

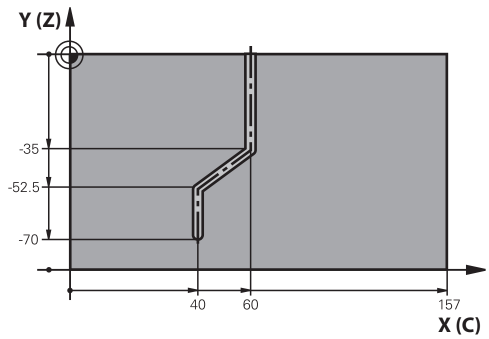


0 BEGIN PGM C27 MM	
1 TOOL CALL 1 Z S2000	Tool call, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y0 R0 FMAX	Pre-position the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MBMAX FMAX	Positioning
5 CYCL DEF 14.0 CONTOUR GEOMETRY	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 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

- 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



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 GEOMETRY	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	

12

Fixed Cycles: Contour Pocket with Contour Formula

12.1 SL cycles with complex contour formula

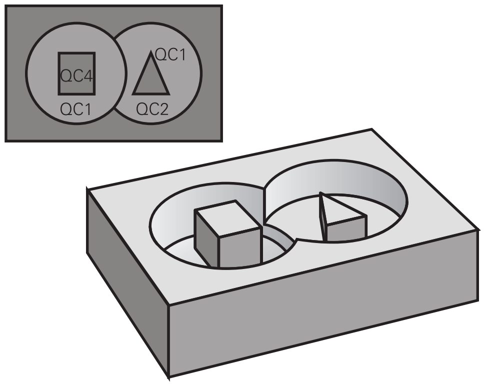
Fundamentals

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.

The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.

To use SL cycles with contour formulas, it is mandatory that your program is structured carefully. These cycles enable you to save frequently used contours in individual NC programs. Using the contour formula, you can connect the subcontours to define a complete contour and specify whether it applies to a pocket or island.

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 KONTUR MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 KONTUR-DATEN ...
8 CYCL DEF 22 RAEUMEN ...
9 CYCL CALL
...
12 CYCL DEF 23 SCHLICHTEN TIEFE ...
13 CYCL CALL
...
16 CYCL DEF 24 SCHLICHTEN SEITE ...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM KONTUR MM

Properties of the subcontours

- The control assumes that each contour is a pocket. Thus, do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are permitted—if they are programmed within the subcontours, they are also effective in the NC programs called subsequently. However, they need not be reset after the cycle call.
- Although the called NC programs can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the NC program.
- Subcontours can be defined with different depths according to your requirements.

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 = “KREIS1“
2 DECLARE CONTOUR  QC2 = “KREISXY“
   DEPTH15
3 DECLARE CONTOUR  QC3 = “DREIECK“
   DEPTH10
4 DECLARE CONTOUR  QC4 =
   “QUADRAT“  DEPTH5
5 QC10 =  ( QC1  |  QC3  |  QC4 ) \ QC2
6 END PGM  MODEL MM
```

```
0 BEGIN PGM  KREIS1 MM
1 CC  X+75 Y+50
2 LP  PR+45 PA+0
3 CP  IPA+360 DR+
4 END PGM  KREIS1 MM
```

```
0 BEGIN PGM  KREIS31XY 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:

Proceed as follows:

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

CONTOUR + POINT MACHINING
- Press the **CONTOUR AND POINT MACHINING** soft key

SEL CONTOUR
- Press the **SEL CONTOUR** soft key.
- Enter the full name of the NC program with the contour definitions

SELECT FILE
- Alternatively, press the **SELECT FILE** soft key and select the program
- Confirm your input with the **END** key

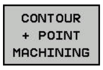
Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR** is no longer necessary if you use **SEL CONTOUR**.

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).

Proceed as follows:

- Press the **SPEC FCT** key



- Press the **CONTOUR AND POINT MACHINING** soft key

- Press the **DECLARE CONTOUR** soft key.
- Enter the number for the contour designator **QC**
- Press the **ENT** key
- Enter the full name of the NC program with the contour descriptions and confirm with the **ENT** key.

- Alternatively, press the **SELECT FILE** soft key and select the NC program
- Define a separate depth for the selected contour
- Press the **END** key

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).

The control will only take different depths (**DEPTH**) into account if the elements overlap. In case of pure islands inside a pocket, this is not the case. Use a simple contour formula for this purpose.
Further information: "SL cycles with simple contour formula", Page 341

Entering a complex contour formula

You can use soft keys to interlink various contours in a mathematical formula.

Proceed as follows:

SPEC FCT

- Press the **SPEC FCT** key

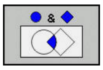
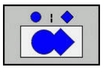
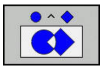
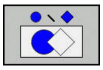
CONTOUR + POINT MACHINING

- Press the **CONTOUR AND POINT MACHINING** soft key

CONTOUR FORMULA

- Press the **CONTOUR FORMULA** soft key
- Enter the number for the contour designator **QC**
- Press the **ENT** key

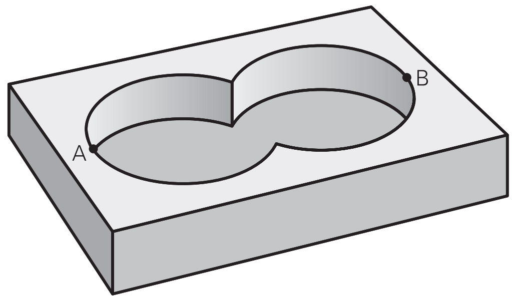
The control displays the following soft keys:

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

> 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
```

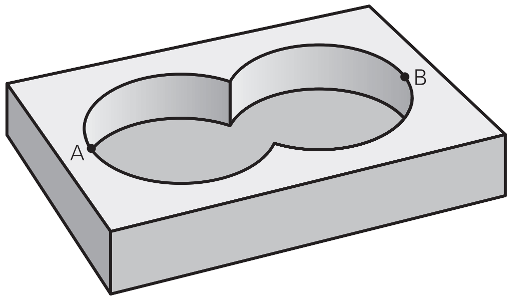
Contour description program 2: pocket B

```
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.



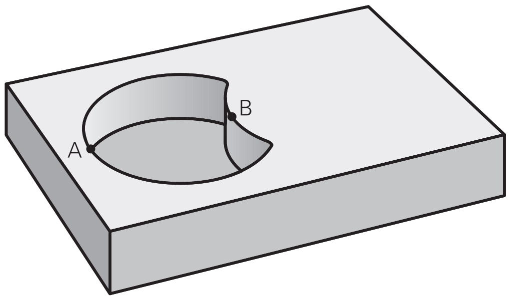
Contour definition program:

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.



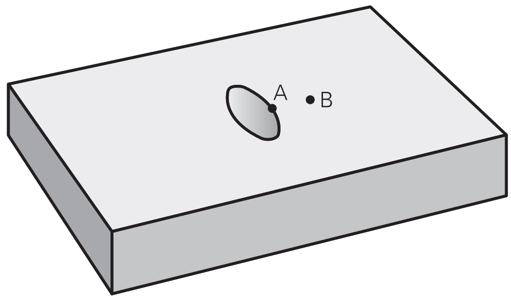
Contour definition program:

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.



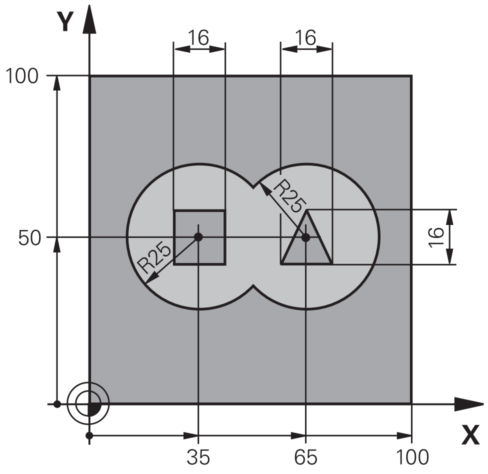
Contour definition program:

```
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 250).

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.	
Q208=+99999 ;RETRACTION FEED RATE	
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	
Q208=+99999 ;RETRACTION FEED RATE	
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	

12.2 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. The control calculates the complete contour from the selected subcontours.

> The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.

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 permitted—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.

Proceed as follows:

SPEC FCT

- ▶ Press the **SPEC FCT** key

CONTOUR + POINT MACHINING

- ▶ Press the **CONTOUR AND POINT MACHINING** soft key

CONTOUR DEF

- ▶ Press the **CONTOUR DEF** soft key
- ▶ Press the **ENT** key
- > The control opens the dialog for entering the contour formula.
- ▶ Enter the first subcontour and confirm with the **ENT** key

POCKET

- ▶ Press the **POCKET** soft key

ISLAND

- ▶ Alternative: Press the **ISLAND** soft key
- ▶ Enter the second subcontour and confirm with the **ENT** key
- ▶ If needed, enter the depth of the second subcontour. Press the **ENT** key
- > Carry on with the dialog as described above until you have entered all subcontours.

You can enter contours in the following ways:

Soft key	Function
CONTOUR <FILE>	Define the name of the contour
SELECT FILE	Alternatively, press the **SELECT FILE** soft key
CONTOUR <FILE>=QS	Define the number of a string parameter
CONTOUR LBL NR	Define the number of a label
CONTOUR LBL NAME	Define the name of the label
CONTOUR LBL QS	Define the number of a string parameter for a label

> 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

> The complete contour is machined with the SL Cycles 20 to 24 (see "Overview", Page 250).

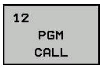
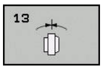
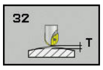
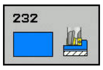
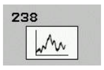
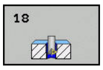
13

Cycles: Special Functions

13.1 Fundamentals

Overview

The control provides the following cycles for the following special purposes:

Soft key	Cycle	Page
	9 DWELL TIME	347
	12 Program call	348
	13 Oriented spindle stop	349
	32 TOLERANCE	350
	225 ENGRAVING of texts	354
	232 FACE MILLING	360
	238 MEASURE MACHINE STATUS	365
	239 ASCERTAIN THE LOAD	367
	18 Thread cutting	369

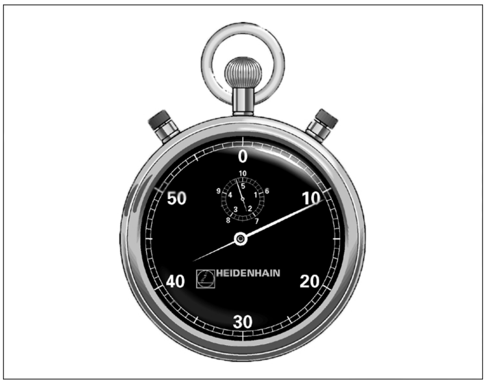
13.2 DWELL TIME (Cycle 9, ISO: G04)

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.

> **i** This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.



Example

89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5

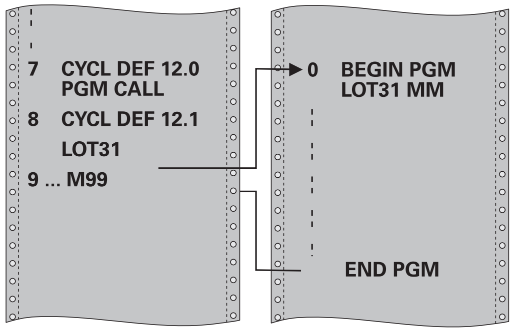
Cycle parameters

- **Dwell time in seconds**: Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

13.3 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:

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

The NC program you are calling must be stored in the internal memory of your control.

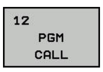
If the NC program you are defining to be a cycle is located in the same directory as the NC program you are calling it from, you need only enter the program name.

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
```

13.4 SPINDLE ORIENTATION (Cycle 13, ISO: G36)

Cycle function

Refer to your machine manual! Machine and control must be specially prepared by the machine tool builder for use of this cycle.

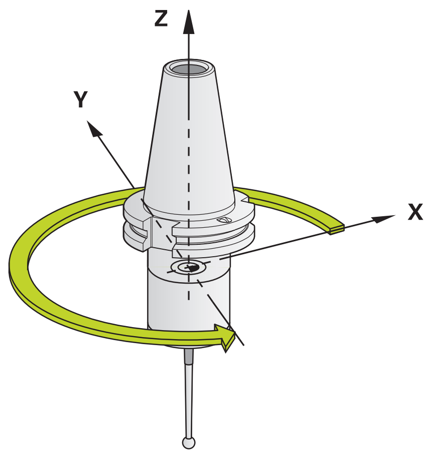
The control can control the main machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for purposes such as:

- 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.



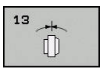
Example

```
93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180
```

Please note while programming:

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes. Cycle 13 is used internally for Cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

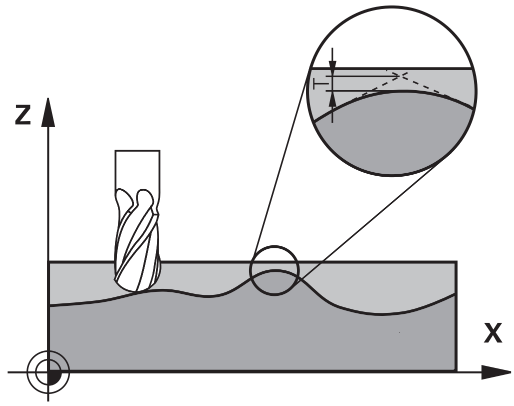


- **Angle of orientation**: Enter the angle with reference to the angle reference axis of the working plane. Input range: 0.0000° to 360.0000°

13.5 TOLERANCE (Cycle 32, DIN/ISO: G62)

Cycle function

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



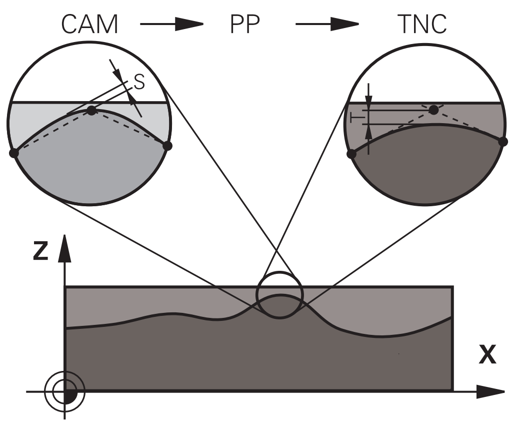
With the entries in Cycle 32 you can influence the result of HSC machining with respect to accuracy, surface definition and speed, inasmuch as the control has been adapted to the machine's characteristics.

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!

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

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.

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.

Reset

The control resets Cycle 32 if you do one of the following:

- Redefine Cycle 32 and confirm the dialog prompt for the **tolerance value** with **NO ENT**.
- Select a new NC program with the **PGM MGT** key.

After you have reset Cycle 32, the control reactivates the tolerance that was predefined by the machine parameters.

Keep the following in mind for 5-axis simultaneous machining!

NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more consistent. Additionally, in this cycle 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

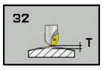
Sample formula for a toroid cutter:

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: Circular constant (pi)
R: Major radius of the torus 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.

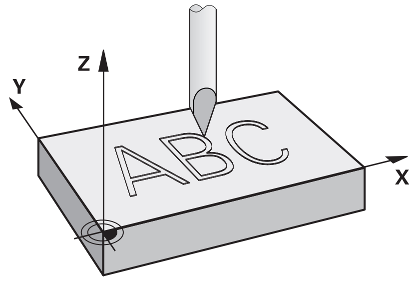
Example

95 CYCL DEF 32.0 TOLERANCE
96 CYCL DEF 32.1 T0.05
97 CYCL DEF 32.2 HSC-MODE:1 TA5

13.6 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:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

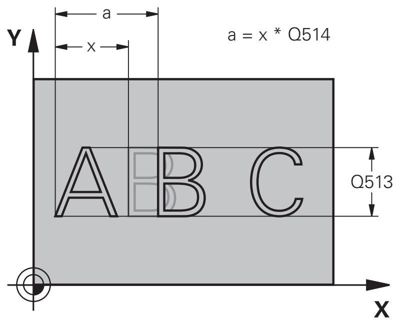
The text to be engraved can also be transferred with a string variable (**QS**).

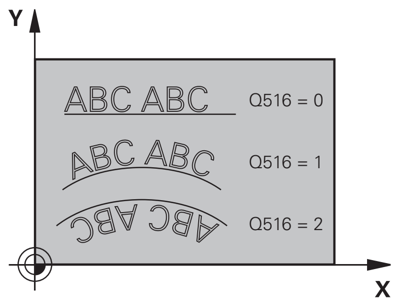
Parameter **Q347** influences the rotational position of the letters.
If **Q374**=0° to 180°, the characters are engraved from left to right.
If **Q374** is greater than 180°, the direction of engraving is reversed.

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 358
- **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**
- **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**





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 MILLING
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

- **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

> 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)

> 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 name and path of an NC program

Use Cycle 225 to engrave the name and path of an NC program.

Define Cycle 225 as usual. Precede the engraved text with **%**.

It is possible to engrave the name or path of an active or called NC program. For this purpose, define **%main<x>** or **%prog<x>**. (Identical to the **ID10010 NR1/2** function)

The following formatting possibilities are available:

Character	Input	Engraved text
Full path of the active NC program	**%main0**	e.g. **TNC:\MILL.h**
Path to the directory of the active NC program	**%main1**	e.g. **TNC:**
Name of the active NC program	**%main2**	e.g. **MILL**
File type of the active NC program	**%main3**	e.g. **.H**
Full path of the called NC program	**%prog0**	e.g. **TNC:\HOUSE.h**
Path to the directory of the called NC program	**%prog1**	e.g. **TNC:**
Name of the called NC program	**%prog2**	e.g. **HOUSE**
File type of the active NC program	**%prog3**	e.g. **.H**

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. operating modes, the control will take the counter reading from the MOD menu into account.

13.7 FACE MILLING (Cycle 232, ISO: G232, software option 19)

Cycle run

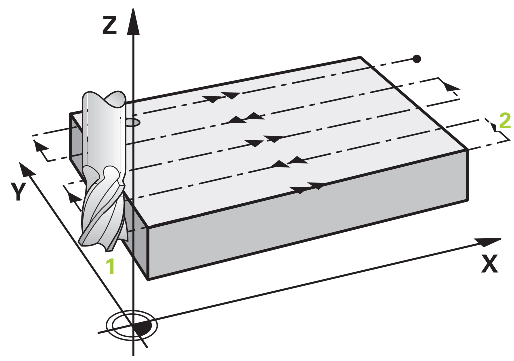
With Cycle 232, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **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.

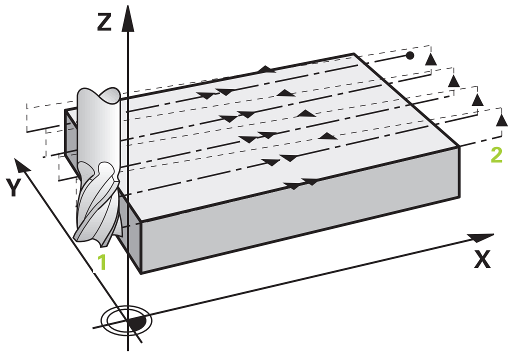
5 The tool then returns to the current infeed depth and moves in the direction of end point **2**

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:

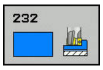
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

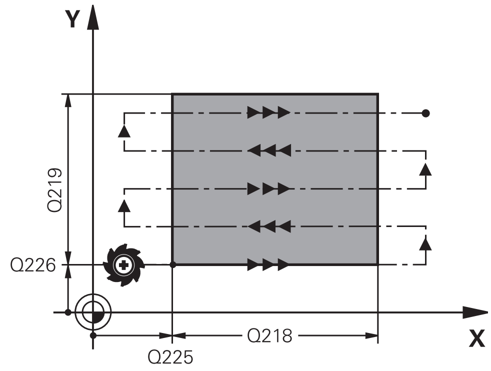
Enter **Q204 2ND SET-UP CLEARANCE** so that no collision with the workpiece or the fixtures can occur.

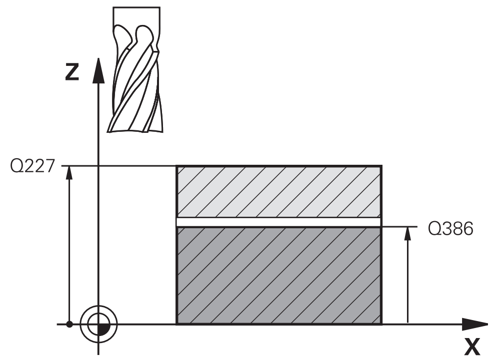
If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).

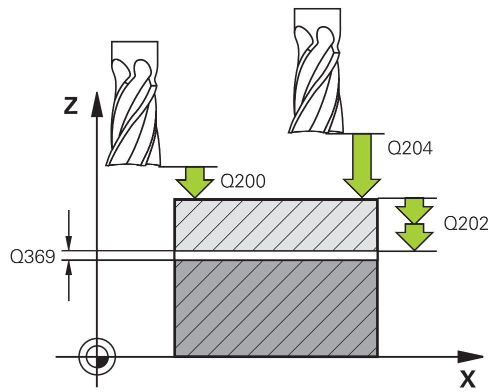
Program **Q227** greater than **Q386**. The control will otherwise display an error message.

Cycle parameters



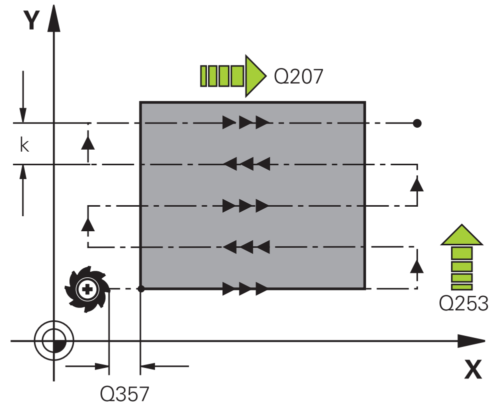






- **Q389 Machining strategy (0/1/2)?**: Determine how the control should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface being machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
  2: Line-by-line 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 principal 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 secondary 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 principal 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 secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referencing 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** influences the following situations:
  Approaching the first plunging depth: **Q357** is the lateral distance from the tool to the workpiece
  Roughing with the milling strategies Q389=0 to 3: The surface to be machined in the **Q350 MILLING DIRECTION** is increased by the value from **Q357** if there is no limit set in this direction
  Side finishing: The paths are extended by the value in **Q357** in the **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 MILLING
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

13.8 MEASURE MACHINE STATUS (Cycle 238, DIN/ISO: G238, option 155)

Application

Refer to your machine manual!

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

Option 155 (**Component Monitoring**) is required for Cycle 238.

During their lifecycle, the machine components which are subject to loads (e.g. guides, ball screws, ...) become worn and thus, the quality of the axis movements deteriorates. This, in turn, affects the production quality.

Using **Component Monitoring** (option 155) and Cycle 238, the control is able to measure the current machine status. Thus, deviations from the shipping condition of the machine that are due to wear and aging can be measured. The measurement results are stored in a text file that is readable for the machine tool builder. He can read the data, evaluate it and react appropriately by scheduling preventive maintenance. This way, unwanted machine downtimes can be prevented!

The machine tool builder can define warning and error thresholds for the measured values and optionally specify error reactions.

Cycle run

Parameter Q570=0

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are effective

Your machine tool builder defines in detail how the axes will move.

Parameter Q570=1

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are **not** effective
3 In the **MON Detail** status tab, you can select the monitoring tasks to be displayed
4 This diagram allows you to watch how close the components are to a warning or error threshold

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

Your machine tool builder defines in detail how the axes will move.

Please note while programming!

NOTICE

Danger of collision!

This cycle may perform extensive movements in one or more axes in rapid traverse! If you program the cycle parameter **Q570**=1, the feed rate and rapid traverse potentiometers, and, if applicable, the spindle potentiometer, have no effect. However, you can stop any movement by setting the feed rate potentiometer to zero. There is a danger of collision!

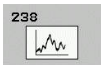
- Before recording measured data, test the cycle in test mode with **Q570=0**
- Contact your machine tool builder to learn about the type and range of movements in Cycle 238 before using the cycle.

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

Cycle 238 is CALL-active.

Ensure that the axes are not clamped before you start the measurement.

Cycle parameters



- **Q570 Mode (0=test/1=measure)?**: Here, you can specify whether the control will perform a measurement of the machine status in test mode or in measurement mode:
  0: No measured data are created. You can control the axis movements with the feed rate and rapid traverse potentiometers
  1: The cycle creates measured data. You **cannot** control the axis movements with the feed rate and rapid traverse potentiometers

Example

```
62 CYCL DEF 238  MEASURE MACHINE
    STATUS
    Q570=+0      ;MODE
```

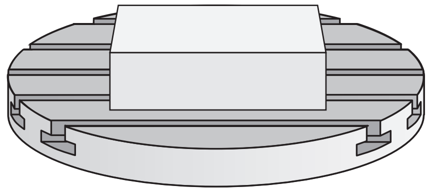
13.9 ASCERTAIN THE LOAD (Cycle 239, DIN/ISO: G239, option 143)

Cycle run

Refer to your machine manual!

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

Option 143 LAC (Load Adaptive Control) is required for Cycle 239.



The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of the table axes. With option 143 LAC (Load Adaptive Control) and Cycle 239 **ASCERTAIN THE LOAD**, the control is able to automatically determine and adjust the actual mass inertia of the load, the actual friction forces and the maximum axis acceleration or reset the feedforward and controller parameters. In this way, you can optimally react to major load changes. The control performs a weighing procedure to ascertain the weight acting on the axes. With this weighing run, the axes move by a specified distance. Your machine tool builder 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 control activates feedforward and, if applicable, controller parameters that allow safe movements of the axis/axes, independently of the current load condition. The parameters set with **Q570**=0 are **independent** of the current load
4 These parameters can be useful during the setup procedure or after the completion of an NC program.

Parameter Q570 = 1

1 The control performs a weighing procedure in which it moves one or more axes. Which axes are moved depends on the setup of the machine and on the drives of the axes.
2 The scope of axis movement is defined by the machine tool builder.
3 The feedforward and controller parameters determined by the control **depend** on the current load.
4 The control activates the ascertained parameters.

Please note while programming:

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.

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.

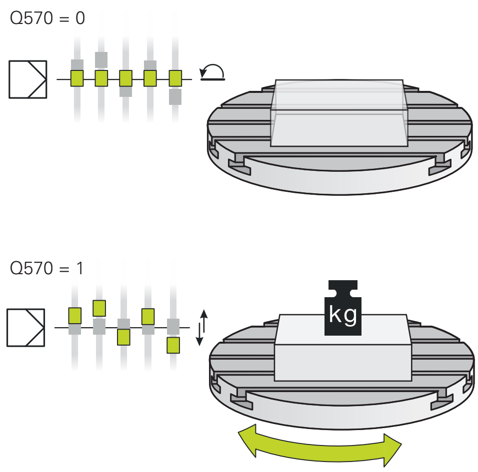
Cycle 239 becomes effective immediately after its definition.

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 a 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.



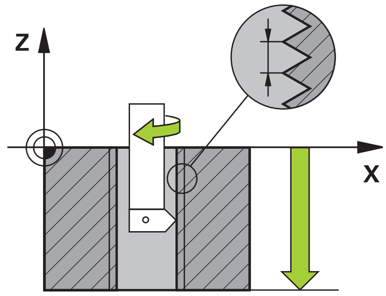
Example

```
62 CYCL DEF 239 ASCERTAIN THE LOAD
    Q570=+0     ;LOAD ASCERTATION
```

13.10 THREAD CUTTING (Cycle 18, DIN/ISO: G86, 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:

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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

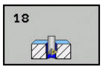
- **sourceOverride** (no. 113603): SpindlePotentiometer (feed rate override is not active) and FeedPotentiometer (speed override is not active); the control then adjusts the spindle speed as required
- **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the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

The spindle speed potentiometer is inactive.

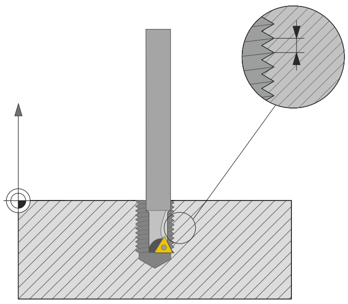
Before calling this cycle, be sure to program a spindle stop! (For example with M5.) The control automatically activates spindle rotation at the start of the cycle and deactivates it at the end.

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

Cycle parameters



- Boring depth (incremental): Enter the thread depth based on the current position. Input range: –99999 to +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

14

Using Touch Probe Cycles

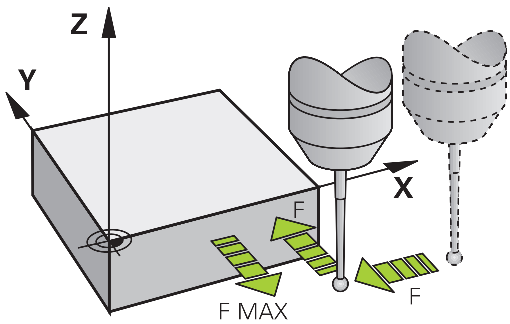
14.1 General information about touch probe cycles

The control must be specifically 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.

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 377

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

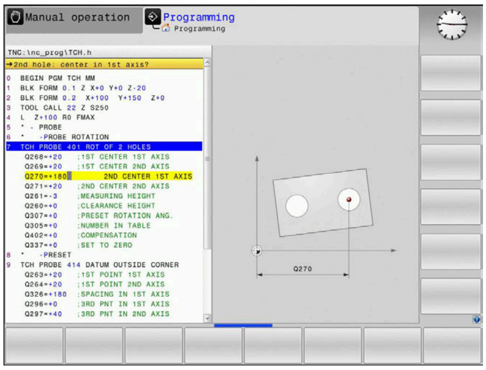
During probing, the control considers an active basic rotation and approaches the workpiece at an angle.

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 Operation and Electronic 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 the Programming mode of operation

Proceed as follows:

TOUCH PROBE
- ▶ Press the **TOUCH PROBE** key.

PRESET
- ▶ Select the desired probing 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, e.g. presetting at pocket center
- > The control opens the programming dialog and prompts you for all required input values. At the same time, a graphic of the input parameters is displayed in the right screen half. The parameter you are prompted for in the dialog is highlighted.
- ▶ Enter all parameters required by the control
- ▶ Confirm each input 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	384
PRESET	Cycles for automatic workpiece presetting	432
MEASURING	Cycles for automatic workpiece inspection	490
SPECIAL CYCLES	Special cycles	536
CALIBRATE TS	Calibrate TS	543
KINEMATICS	Kinematics	559
TT CYCLES	Cycles for automatic tool measurement (enabled by the machine manufacturer)	592

NC blocks

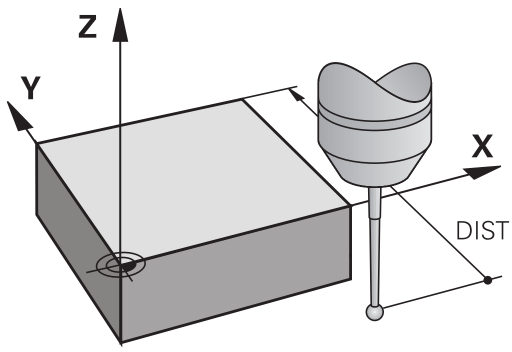
```
5 TCH PROBE 410 PRESET INSIDE
  RECTAN.
  Q321=+50   ;CENTER IN 1ST AXIS
  Q322=+50   ;CENTER IN 2ND AXIS
  Q323=60    ;FIRST SIDE LENGTH
  Q324=20    ;2ND SIDE LENGTH
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q301=0     ;MOVE TO CLEARANCE
  Q305=10    ;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
```

14.2 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.

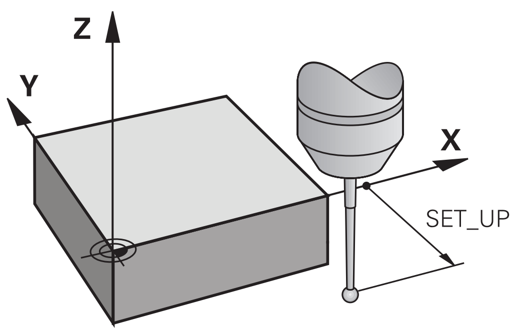
Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the range defined in **DIST**, the control will issue an error message.



Set-up clearance to touch point: SET_UP in touch probe table

In **SET_UP** you define how far from the defined (or calculated) touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to **SET_UP**.



Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table

To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.

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 be greater than set in the optional **maxTouchFeed** machine parameter (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**, Cycle **11 SCALING**, and Cycle **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.

Depending on the setting of the optional **chkTiltingAxes** machine parameter (no. 204600), the control will check during probing whether the position of the rotary axes matches the tilting angles (3D-ROT). If that is not the case, the control displays an error message.

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.

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.

14.3 Touch-probe table

General information

Various data are 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

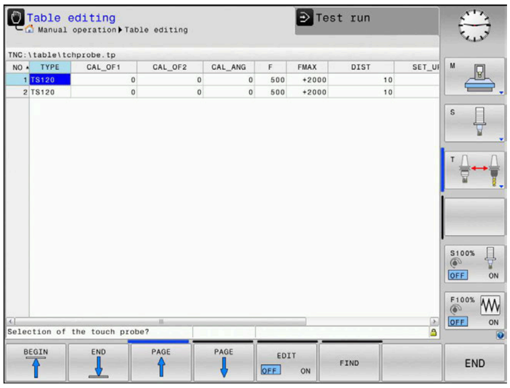
Proceed as follows:

- Press the **Manual operation** key

- Press the **TOUCH PROBE** soft key
- > The control displays additional soft keys.

- 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 entries	–
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 secondary 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 be greater than set in the optional **maxTouch-Feed** machine parameter (no. 122602).	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe is pre-positioned 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 for 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?**

Abbr.	Inputs	Dialog
REACTION	As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. The entry defines how the control is to react to the resetting of the ready signal ■ **NCSTOP**: The NC program will be interrupted ■ **EMERGSTOP**: Emergency stop, quick braking of the axes	**Reaction?**

With **TS 642** touch probes, you can select between **TS642-3** and **TS642-6** in the **TYPE** column. Values 3 and 6 correspond to the switch settings in the battery case of the touch probe.

- **3**: Activation of the touch probe via a switch in the taper shank. Do not use this mode. It is not supported by HEIDENHAIN controls yet.
- **6**: Activation of the touch probe via an infrared signal. Select this mode.

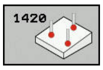
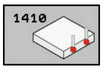
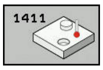
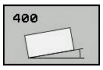
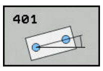
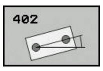
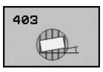
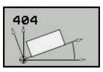
15

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

15.1 Overview

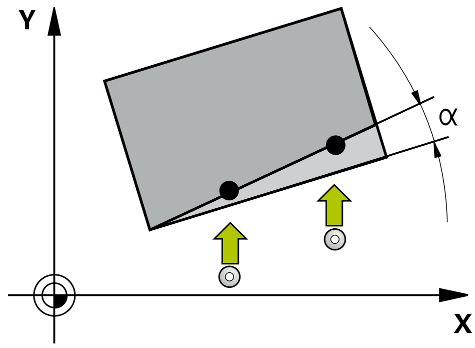
The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Cycle	Page
	1420 PROBING IN PLANE Automatic measurement using three points. Compensation via basic rotation.	393
	1410 PROBING ON EDGE Automatic measurement using two points. Compensation via basic rotation or a rotation of the rotary table.	397
	1411 PROBING TWO CIRCLES Automatic measurement using two cylindrical holes or studs. Compensation via basic rotation or a rotation of the rotary table.	401
	400 BASIC ROTATION Automatic measurement using two points. Compensation via basic rotation.	407
	401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.	410
	402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.	414
	403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation by turning the table.	419
	405 ROT IN C AXIS Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.	424
	404 SET BASIC ROTATION Setting any basic rotation.	428

15.2 Touch probe cycles 14xx: fundamentals

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 probing positions reference the programmed nominal positions in the I-CS.
> See the drawing for these nominal positions.
> Before a cycle definition you must program a tool call to define the touch-probe axis.

Explanation of terms

Designation	Short description
Nominal position	Position in the drawing, e.g. position of a hole
Nominal dimension	Dimension in the drawing, e.g. hole diameter
Actual position	Measured position, e.g. position of a hole
Actual dimension	Measured dimension, e.g. hole diameter
I-CS	I-CS: **Input Coordinate System**
W-CS	W-CS: **Workpiece Coordinate System**
Object	Object to be probed: circle, stud, plane, edge

Evaluation – preset:

- If you want to probe objects in a consistent machining plane or probe objects while TCPM is active, you can program any required shifts as basic transformations in the preset table.
- 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.

> When probing, existing 3-D calibration data are taken into account. If these calibration data do not exist, deviations might be the result.
>
> If you want to use not only the measured rotation, but also a measured position, make sure to probe the surface perpendicularly, if possible. 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.

Logging:

The determined results are recorded in the **TCHPRAUTO.html** file and stored in the Q parameters programmed for this cycle.
The measured deviations are the differences between the measured actual values and the mean tolerance value. If no tolerance has been specified, they refer to the nominal dimension.

Semi-automatic mode

If the probing positions relative to the current datum are unknown, you can execute the cycle in semi-automatic mode. In this mode, you can determine the starting position by manually pre-positioning before performing the probing operation on the desired object.

For this purpose, precede the value for the required nominal position with **"?"**. This can be done via the **ENTER TEXT** soft key. Depending on the object, you need to define the nominal positions that determine the probing direction, see "Examples".

Cycle run:

1 The cycle interrupts the NC program.

2 A dialog window opens

Proceed as follows:

- ▶ Use the axis-direction keys to pre-position the touch probe at the desired point
- ▶ Alternatively, use the handwheel for pre-positioning
- ▶ If required, change the probing conditions, such as the probing direction.
- ▶ Press **NC start**
- > If you programmed value 1 or 2 for **Q1125** Traverse to clearance height, then the control will open a pop-up window, explaining that the Traverse to clearance height mode cannot be used here.
- ▶ While the pop-up window is still open, use the axis keys to move to a safe position
- ▶ Press **NC start**
- > Program execution is resumed.

NOTICE

Danger of collision!

The control will ignore the programmed values 1 or 2 for Traverse to clearance height when running in semi-automatic mode. Depending on the position of the touch probe, there is danger of collision.

- ▶ In semi-automatic mode, traverse to clearance height after each probing operation.

See the drawing for these nominal positions.

Semi-automatic mode can only be executed in the machine operating modes, i.e. not in the Test run operating mode.

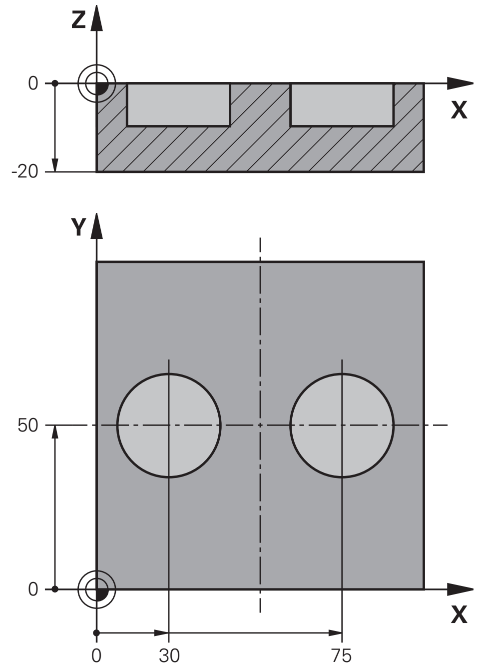
If you did not define a nominal position for a touch point in any direction, the control generates an error message.

If you did not define a nominal position for a single direction, the control will capture the actual position after probing the object. This means that the measured actual position will subsequently be applied as the nominal position. Consequentially, there is no deviation for this position and thus no position compensation.

Examples

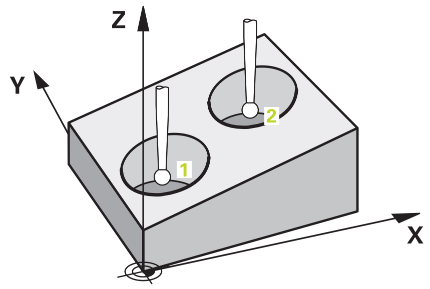
Important: Specify the **nominal positions** from the drawing!

In the following three examples, the nominal positions from this drawing will be used.



Hole

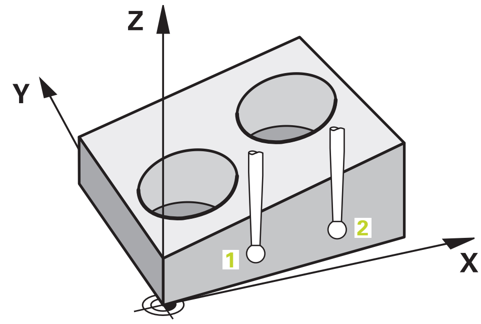
In this example, you will align two holes. Probing is done in the X axis (principal axis) and in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure in this direction.



5 TCH PROBE 1411 PROBING TWO CIRCLES		Define the cycle
QS1100= "?30"	;1ST POINT REF AXIS	Nominal position 1 of principal axis exists, position of the workpiece unknown
QS1101= "?50"	;1ST POINT MINOR AXIS	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?"	;1ST POINT TOOL AXIS	Nominal position 1 of tool axis unknown
Q1116=+10	;DIAMETER 1	Diameter at 1st position
QS1103= "?75"	;2ND POINT REF AXIS	Nominal position 2 of principal axis exists, position of the workpiece unknown
QS1104= "?50"	;2ND POINT MINOR AXIS	Nominal position 2 of secondary axis exists, position of workpiece unknown
QS1105= "?"	;2ND POINT TOOL AXIS	Nominal position 2 of tool axis unknown
Q1117=+10	;DIAMETER 2	Diameter at 2nd position
Q1115=+0	;GEOMETRY TYPE	Geometry type: two holes
...	;	

Edge

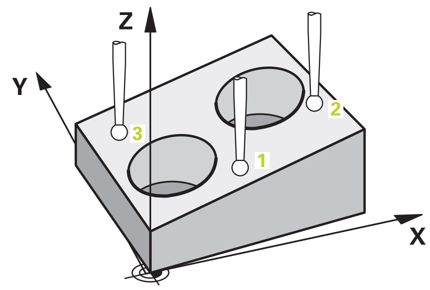
In this example, you will align an edge. Probing is on done in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for this axis! Nominal positions for the X axis (principal axis) and for the Z axis (tool axis) are not required because you will not measure in these directions.



5 TCH PROBE 1410 PROBING ON EDGE		Define the cycle
QS1100= "?"	**;1ST POINT REF AXIS**	Nominal position 1 of principal axis unknown
QS1101= "?0"	**;1ST POINT MINOR AXIS**	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?"	**;1ST POINT TOOL AXIS**	Nominal position 1 of tool axis unknown
QS1103= "?"	**;2ND POINT REF AXIS**	Nominal position 2 of principal axis unknown
QS1104= "?0"	**;2ND POINT MINOR AXIS**	Nominal position 2 of secondary axis exists, position of workpiece unknown
QS1105= "?"	**;2ND POINT TOOL AXIS**	Nominal position 2 of tool axis unknown
Q372=+2	**;PROBING DIRECTION**	Probing direction Y+
...	**;**	

Plane

In this example, you will align a plane. In this case, it is mandatory to define all three nominal positions. For angle calculations, it is important that all three axes are taken into account when probing.



5 TCH PROBE 1420 PROBING IN PLANE	Define the cycle
QS1100= "?50" ;1ST POINT REF AXIS	Nominal position 1 of principal axis exists, position of the workpiece unknown
QS1101= "?10" ;1ST POINT MINOR AXIS	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?0" ;1ST POINT TOOL AXIS	Nominal position 1 of tool axis exists, position of the workpiece unknown
QS1103= "?80" ;2ND POINT REF AXIS	Nominal position 2 of principal axis exists, position of the workpiece unknown
QS1104= "?50" ;2ND POINT MINOR AXIS	Nominal position 2 of secondary axis exists, position of workpiece unknown
QS1105= "?0" ;2ND POINT TOOL AXIS	Nominal position 2 of tool axis exists, position of the workpiece unknown
QS1106= "?20" ;3RD POINT REF AXIS	Nominal position 3 of principal axis exists, position of the workpiece unknown
QS1107= "?80" ;3RD POINT MINOR AXIS	Nominal position 3 of secondary axis exists, position of workpiece unknown
QS1108= "?0" ;3RD POINT TOOL AXIS	Nominal position 3 of tool axis exists, position of the workpiece unknown
Q372=-3 ;PROBING DIRECTION	Probing direction Z-
... ;	

Evaluation of tolerances

Optionally, the control can monitor the cycles for tolerances. This includes monitoring the position and size 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. The preset will be corrected only afterwards, if required.

Cycle run:

- With an error reaction parameter set to **Q309**=1, the control will check for scrap and rework. If you defined **Q309**=2, the control will only check for scrap.
- If the actual position determined by probing is erroneous, the control will interrupt execution of the NC program. A dialog window opens, showing all nominal and actual dimensions of the object
- You can then decide whether to continue machining or abort the NC program. To resume NC program operation, press **NC start**. To abort the program, press the **CANCEL** soft key

> 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, the control will save the values with respect to the mean tolerance in the programmed Q parameters. You can further process them.

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.

For this purpose, enter **"@"**after the value for the required nominal position. This can be done via the **ENTER TEXT** soft key. Enter the actual position after the "**@**".

> If you program @, no probing will be carried out.
> The control only accounts for the actual and nominal positions.
>
> You must define the actual position for all three axes: principal axis, secondary axis, and tool axis. If you define only one axis with its actual position, an error message will be generated.
>
> Actual positions can also be defined with Q parameters **Q1900-Q1999**.

Example:

This feature allows you to do the following:

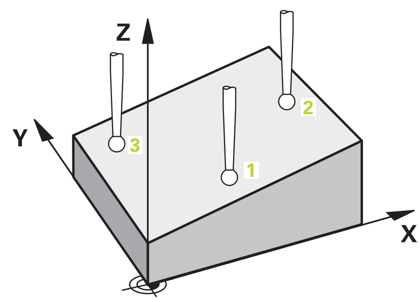
- Determine a circular pattern based on multiple different objects
- Align a gear wheel via its center and the position of a tooth

5 TCH PROBE 1410 PROBING ON EDGE	
QS1100= "10+0.02@10.0123"	
;1ST POINT REF AXIS	Nominal position 1 of the principal axis with tolerance monitoring and actual position
QS1101="50@50.0321"	
;1ST POINT MINOR AXIS	Nominal position 1 of the secondary axis with tolerance monitoring 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
... ;	

15.3 PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420, 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"), at 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 If you programmed Traverse to clearance height, the touch probe returns to clearance height (depending on the setting in **Q1125**). It then moves in the working plane to touch point **2** to measure there the actual value of the second touch point in 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 position 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 W-CS
Q980 to **Q982**	Deviation 1 of the positions
Q983 to **Q985**	Deviation 2 of the positions
Q986 to **Q988**	Deviation 3 of the positions
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

- Always move to clearance height between objects or touch points

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

HEIDENHAIN recommends avoiding the use of axis angles in this cycle!

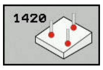
The control can only calculate the angular values if the three touch points are not positioned on a straight line.

The nominal spatial angle results from the defined nominal positions. The control saves the calculated spatial angle in parameters **Q961** to **Q963**. The control transfers the difference between the measured spatial angle and the nominal spatial angle to the basic rotation in 3-D of the preset table.

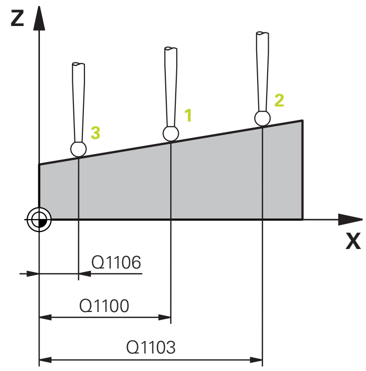
Aligning the rotary table axes:

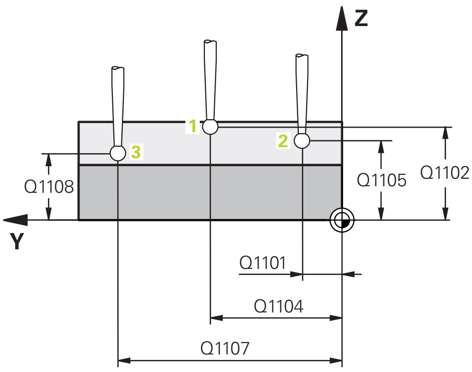
- Alignment with rotary table axes is only possible if two rotary table axes have been defined in the kinematics.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible to align the rotary table axes without defining evaluation of the rotation.

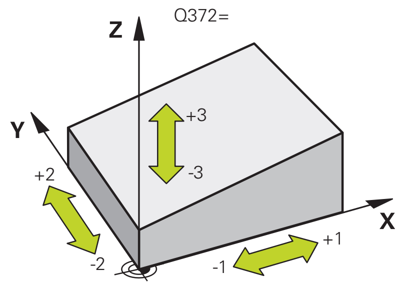
Cycle parameters

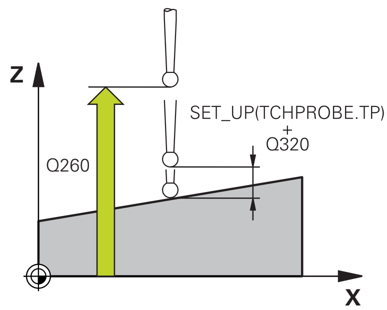


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal position 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 position of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): nominal position 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 position of the third touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1107 3rd noml. position minor axis?** (absolute): Nominal position of the third touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1108 3rd nominal position tool axis?** (absolute): Nominal position 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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
- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the touch 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 object
  2: Move to clearance height before and after moving to each touch point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the 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 touch point will be used to correct the current preset:
  0: No correction
  1: Correction based on the 1st touch point
  2: Correction based on the 2nd touch point
  3: Correction based on the 3rd touch point
  4: Correction based on the averaged touch point position
- **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.

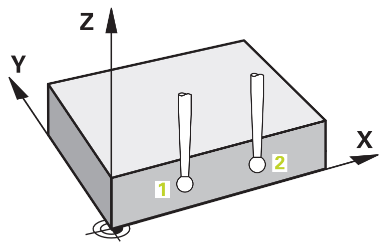
Example

5 TCH PROBE 1420 PROBING IN PLANE	
Q1100=+0	;1ST POINT REF AXIS
Q1101=+0	;1ST POINT MINOR AXIS
Q1102=+0	;1ST POINT TOOL AXIS
Q1103=+0	;2ND POINT REF AXIS
Q1104=+0	;2ND POINT MINOR AXIS
Q1105=+0	;2ND POINT TOOL AXIS
Q1106=+0	;3RD POINT REF AXIS
Q1107=+0	;3RD POINT MINOR AXIS
Q1108=+0	;3RD POINT MINOR AXIS
Q372=+1	;PROBING DIRECTION
Q320=+0	;SET-UP CLEARANCE
Q260=+100	;CLEARANCE HEIGHT
Q1125=+2	;CLEAR. HEIGHT MODE
Q309=+0	;ERROR REACTION
Q1126=+0	;ALIGN ROTARY AXIS
Q1120=+0	;TRANSER POSITION
Q1121=+0	;CONFIRM ROTATION

15.4 PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410, option 17)

Cycle run

Touch probe cycle 1410 determines the misalignment of a workpiece by measuring two points located on an edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic ("Executing touch probe cycles"), at 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 I-CS
Q965	Measured angle of rotation in the coordinate system of the rotary table
Q980 to **Q982**	Deviation 1 of the positions
Q983 to **Q985**	Deviation 2 of the positions
Q994	Measured angle deviation in the I-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)

Please note while programming!

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

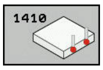
- Always move to clearance height between objects or touch points

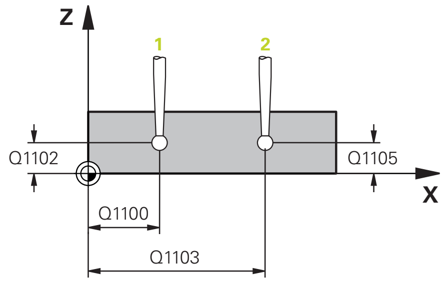
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

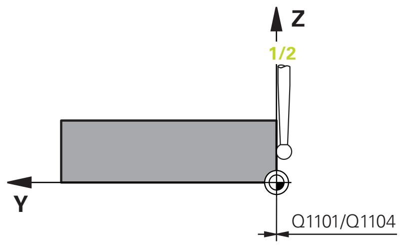
Aligning the rotary table axes:

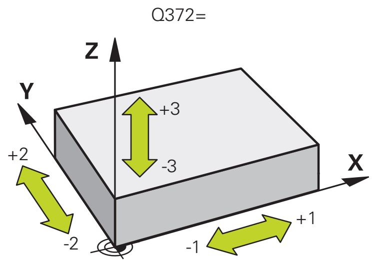
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible to align the rotary table axes and activate the basic rotation at the same time

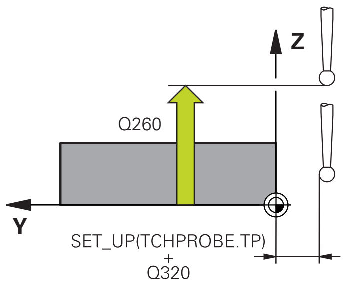
Cycle parameters











- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal position 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 position of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): nominal position 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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

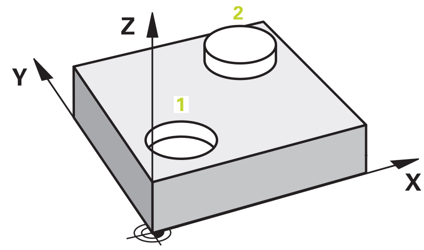
- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the touch points:
  –1: Do not move to clearance height
  0: Move to clearance height before and after the cycle
  1: Move to clearance height before and after moving to each object
  2: Move to clearance height before and after moving to each touch point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the 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 touch point will be used to correct the current preset:
  0: No correction
  1: Correction based on the 1st touch point
  2: Correction based on the 2nd touch point
  3: Correction based on the averaged touch point position
- **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

15.5 PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411, option 17)

Cycle run



Touch probe cycle 1411 captures the center points of two holes or cylindrical studs and calculates a straight line connecting these center points. The cycle determines the rotation in the working plane based on the difference between the measured angle and the nominal angle.

1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic ("Executing touch probe cycles"), at 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 I-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**	Deviation 1 of the positions
Q983 to **Q985**	Deviation 2 of the positions
Q994	Measured angle deviation in the I-CS
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!

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

- Always move to clearance height between objects or touch points

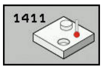
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

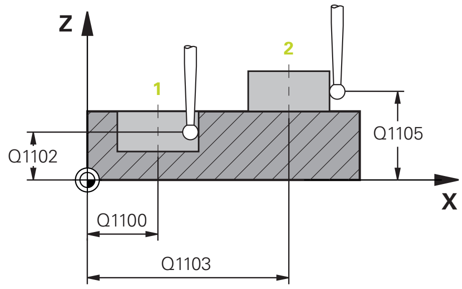
If the hole is so small that the programmed set-up clearance cannot be met, a dialog opens. It shows the nominal dimension of the hole, the calibrated ball-tip radius, and the achievable set-up clearance.
Either confirm the dialog with **NC start** or cancel the process via a soft key. If you confirm with **NC start**, the effective set-up clearance will be reduced to the displayed value, but only for this object.

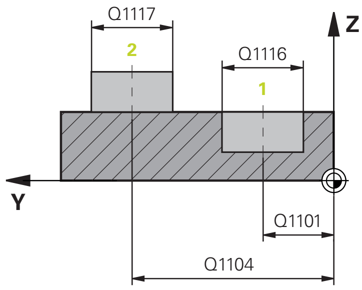
Aligning the rotary table axes:

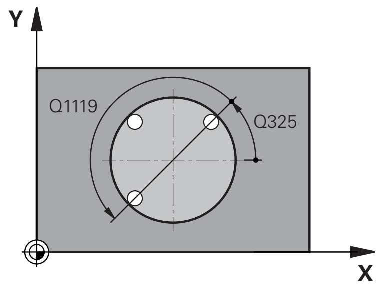
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible to align the rotary table axes and activate the basic rotation at the same time

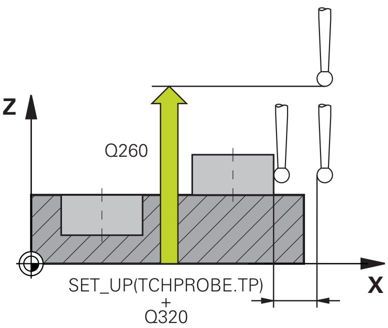
Cycle parameters











- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal position 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 position of the second touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): nominal position 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
  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 touch points on the diameter. Input range: 3 to 8
- **Q325 Starting angle?** (absolute): Angle between the principal 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 +360.000
- **Q320 Set-up clearance?** (incremental): Additional distance between the touch 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 touch 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 object
  2: Move to clearance height before and after moving to each touch point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the 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 touch point will be used to correct the current preset:
  0: No correction
  1: Correction based on the 1st touch point
  2: Correction based on the 2nd touch point
  3: Correction based on the averaged touch point position

Example

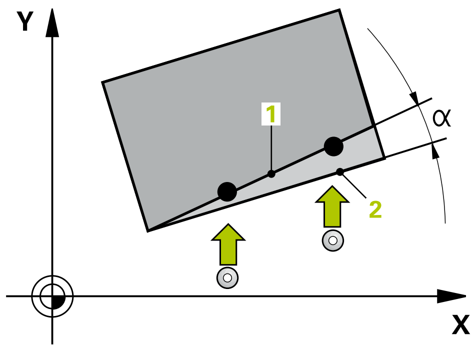
5 TCH PROBE 1410 PROBING TWO CIRCLES	
Q1100=+0	;1ST POINT REF AXIS
Q1101=+0	;1ST POINT MINOR AXIS
Q1102=+0	;1ST POINT TOOL AXIS
Q1116=0	;DIAMETER 1
Q1103=+0	;2ND POINT REF AXIS
Q1104=+0	;2ND POINT MINOR AXIS
Q1105=+0	;2ND POINT TOOL AXIS
Q1117=+0	;DIAMETER 2
Q1115=0	;GEOMETRY TYPE
Q423=4	;NO. OF PROBE 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

- **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

15.6 Touch probe cycles 4xx: fundamentals

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**.

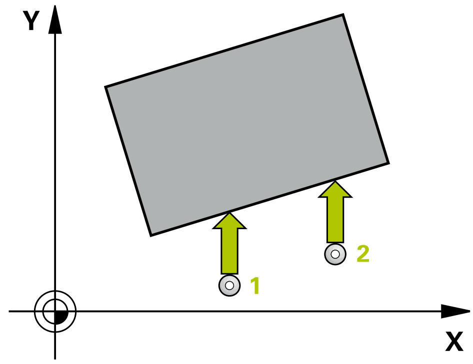


> These cycles do not work with 3-D Rot! In such a case, use Cycles 14xx. **Further information:** "Touch probe cycles 14xx: fundamentals", Page 385

15.7 BASIC ROTATION (Cycle 400, DIN/ISO: G400, 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 379), 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:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

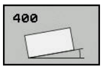
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

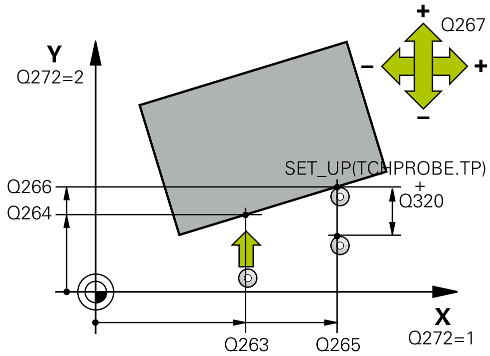
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

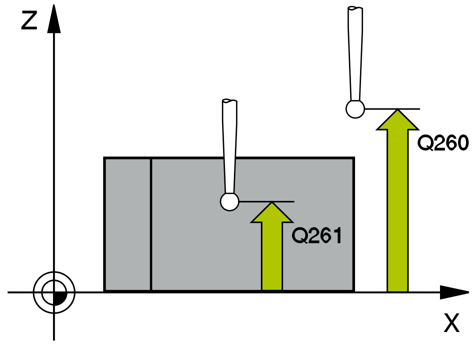
The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal 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 secondary 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 principal 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 secondary 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: Principal axis = measuring axis
  2: Secondary 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): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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





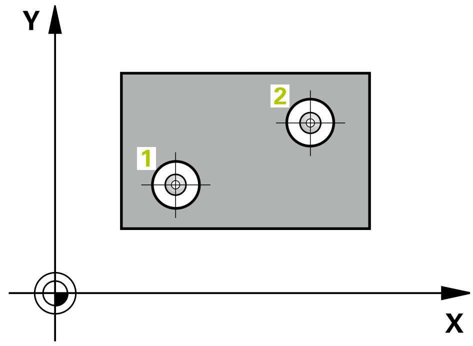
Example

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**

- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the principal 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 stores the determined basic rotation in the ROT menu of the Manual Operation mode. Input range: 0 to 99999

15.8 ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401, 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 379), 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:

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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

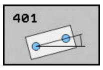
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control will reset an active basic rotation at the beginning of the cycle.

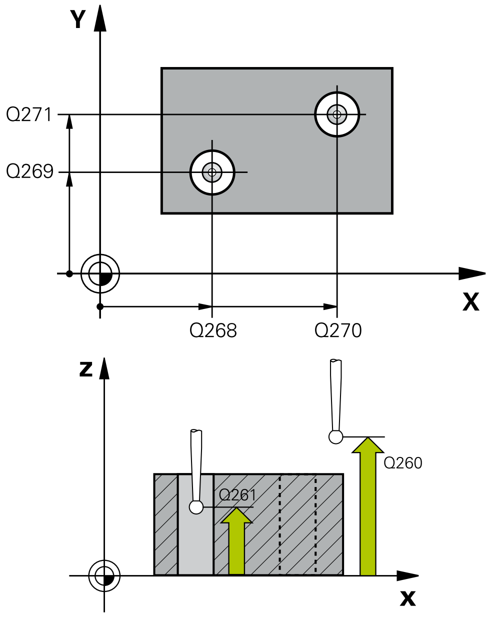
If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **Q268 1st hole: center in 1st axis?** (absolute): Center of the first hole in the principal 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 secondary 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 principal 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 secondary 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

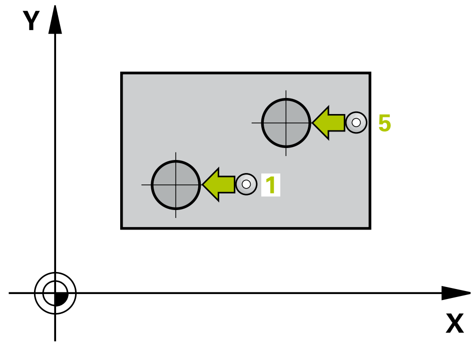
- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the principal axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line 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 99 999
  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**

Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+20	;CLEARANCE HEIGHT
Q307=0	;PRESET ROTATION ANG.
Q305=0	;NUMBER IN TABLE
Q402=0	;COMPENSATION
Q337=0	;SET TO ZERO

15.9 ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402, 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 379), 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:

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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

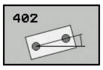
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control will reset an active basic rotation at the beginning of the cycle.

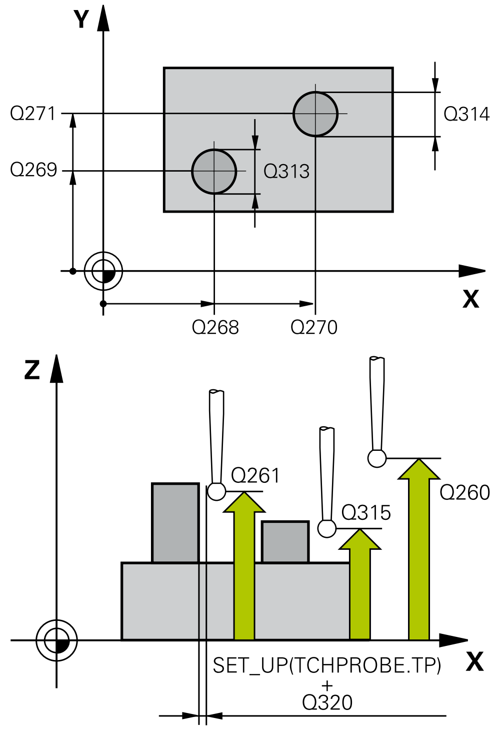
If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X

Cycle parameters



- **Q268 1st stud: center in 1st axis?** (absolute): Center of the first stud in the principal 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 secondary 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 principal 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 secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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

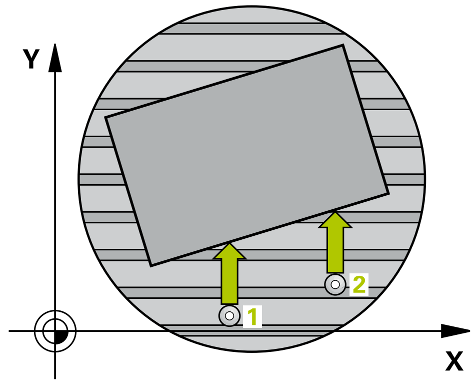
5 TCH PROBE 402 ROT OF 2 STUDS	
Q268=-37	;1ST CENTER 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q313=60	;DIAMETER OF STUD 1
Q261=-5	;MEAS. HEIGHT STUD 1
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q314=60	;DIAMETER OF STUD 2
Q315=-5	;MEAS. HEIGHT STUD 2
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE 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 principal 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 99 999
  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**

15.10 ROT. IN ROTARY AXIS (Cycle 403, DIN/ISO: G403, 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 379), 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 probing 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!

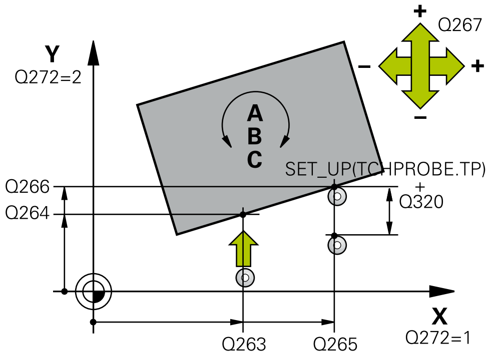
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

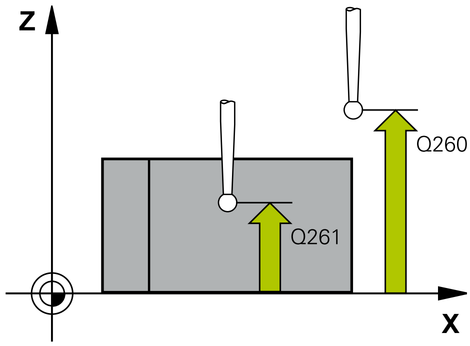
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal 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 secondary 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 principal 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 secondary 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: Principal axis = measuring axis
  2: Secondary 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): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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

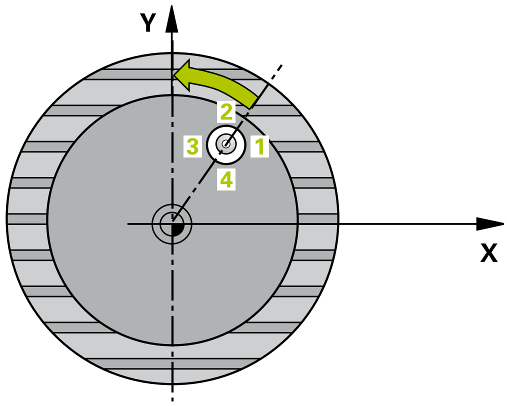
5 TCH PROBE 403 ROT IN ROTARY AXIS	
Q263=+0	;1ST POINT 1ST AXIS
Q264=+0	;1ST POINT 2ND AXIS
Q265=+20	;2ND PNT IN 1ST AXIS
Q266=+30	;2ND POINT 2ND AXIS
Q272=1	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q312=0	;COMPENSATION AXIS
Q337=0	;SET TO ZERO
Q305=1	;NUMBER IN TABLE
Q303=+1	;MEAS. VALUE TRANSFER
Q380=+90	;REFERENCE ANGLE

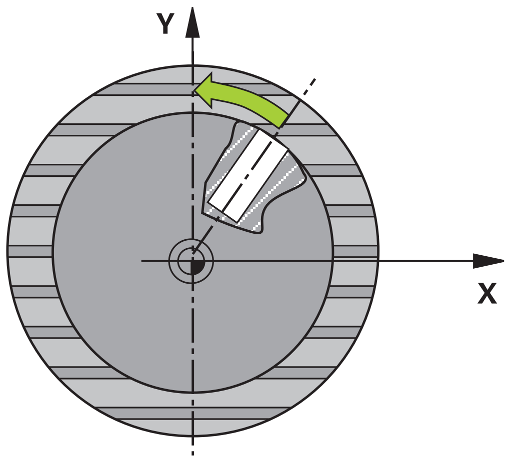
- **Q312 Axis for compensating movement?**:
  Specify the rotary axis in which the control is to compensate the measured misalignment:
  0: Automatic mode—the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed 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: 0 to 360.000

15.11 ROT. IN C AXIS (Cycle 405, DIN/ISO: G405, 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
- the angular offset between the nominal position and the actual position of a hole center point

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 "Executing touch probe cycles", at 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 point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in parameter **Q150**.

Please note while programming:

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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

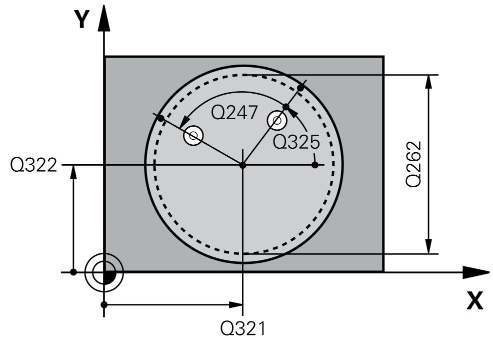
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

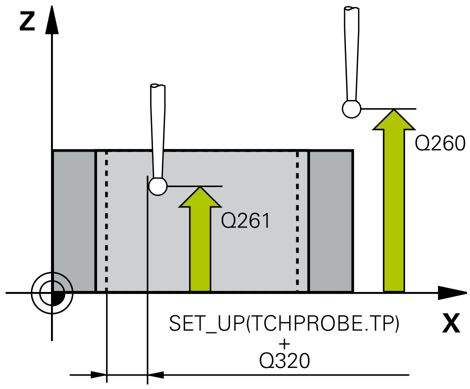
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The smaller the stepping angle, the less accurately the control can calculate the circle center point. Minimum input value: 5°.

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the hole in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the hole in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point with the positive Y axis. If you program **Q322** not equal to zero 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 principal 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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.

15.12 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404, 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.

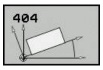
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Example

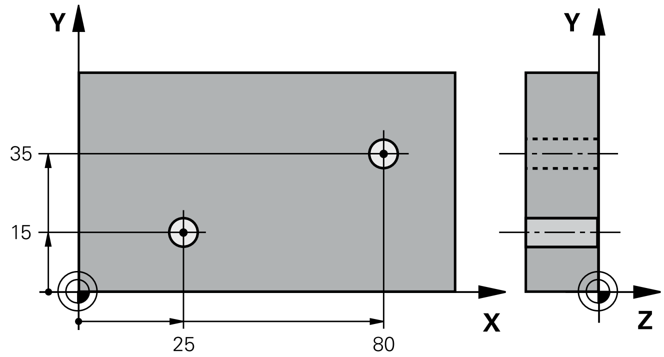
```
5 TCH PROBE 404  SET BASIC ROTATION
    Q307=+0      ;PRESET ROTATION ANG.
    Q305=-1      ;NUMBER IN TABLE
```

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 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.

15.13 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	

16

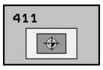
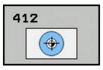
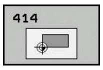
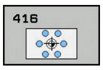
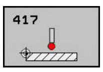
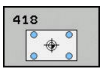
Touch Probe Cycles: Automatic Presetting

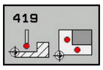
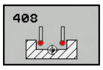
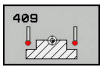
16.1 Fundamentals

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
	410 DATUM INSIDE RECTANGLE Measuring the inside length and width of a rectangle, and defining the center as preset	435
	411 DATUM OUTSIDE RECTANGLE Measuring the outside length and width of a rectangle, and defining the center as preset	439
	412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as preset	443
	413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as preset	448
	414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as preset	453
	415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as preset	458
	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	463
	417 PRESET IN TS AXIS (2nd soft-key row) Measuring any position in the touch probe axis and defining it as preset	467
	418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as preset	469

Soft key	Cycle	Page
	419 DATUM IN ONE AXIS (2nd soft-key row) Measuring any position in any axis and defining it as preset	474
	408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as preset	477
	409 RIDGE CENTER PRESET Measuring the outside width of a ridge, and define the ridge center point as the preset	481

The control must be specifically 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.

Characteristics common to all touch probe cycles for presetting

> 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 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, deleting 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 result is written to the preset table, row **Q305**, the machine coordinate system is the reference system (REF coordinates); **activate the preset using Cycle 247 in the NC program**
- **Q305 not equal to 0, Q303 = -1**

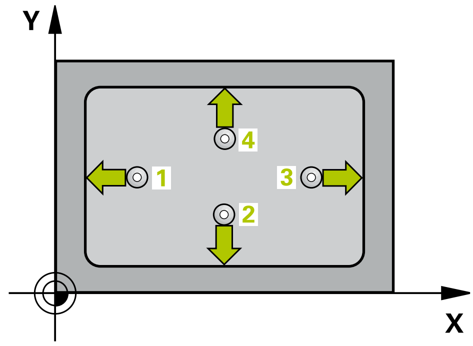
> This combination can only occur if you
> - read in NC programs containing Cycles 410 to 418 created on a TNC 4xx
> - read in NC programs containing Cycles 410 to 418 created with an older software version on an iTNC 530
> - did not specifically define the measured-value transfer with parameter **Q303** 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.

16.2 PRESET INSIDE RECTANGLE (Cycle 410, DIN/ISO: G410, 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 "Executing touch probe cycles", at 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 434)
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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

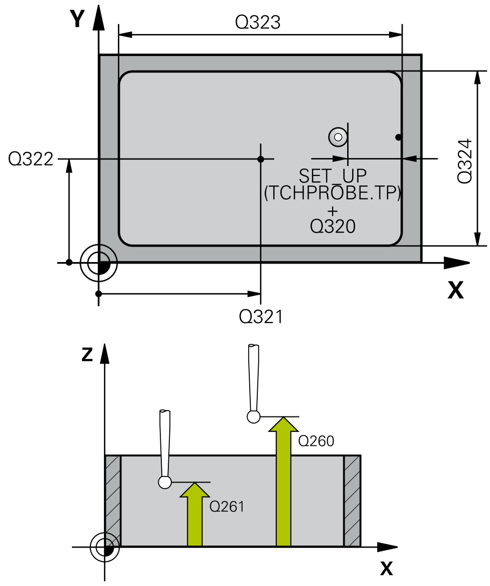
To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides. If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center 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.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the pocket in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q323 First side length?** (incremental): Pocket length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q324 Second side length?** (incremental): Pocket length, parallel to the secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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 principal axis at which the control should set the pocket center. Default setting = 0. Input range: –99999.9999 to 99999.9999



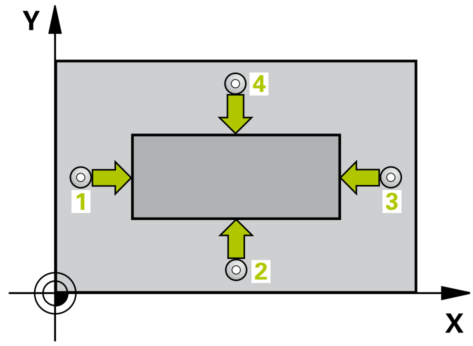
Example

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 secondary 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 434)
  0: Write the determined 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 principal 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 secondary 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

16.3 PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411, 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 379), at 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 434)
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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

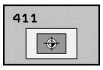
Danger of collision!

To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

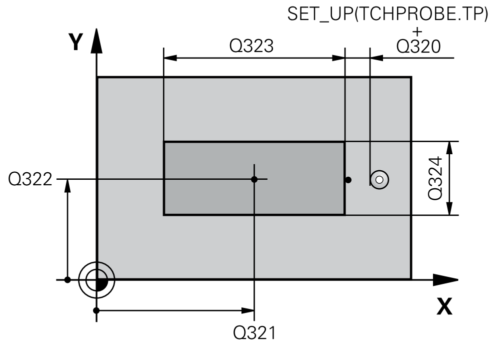
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

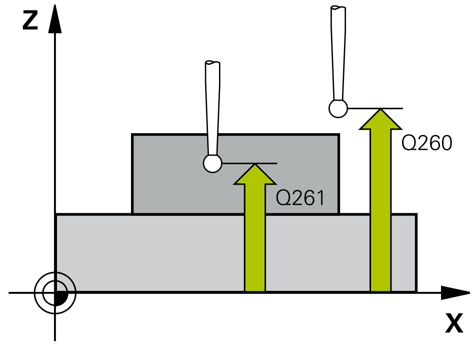
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q323 First side length?** (incremental): Stud length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q324 Second side length?** (incremental): Stud length, parallel to the secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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 principal 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 secondary axis at which the control should set the stud center. Default setting = 0. Input range: –99999.9999 to 99999.9999





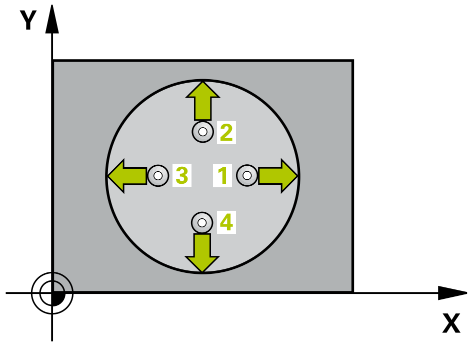
Example

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 434)
  0: Write the determined 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 principal 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 secondary 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

16.4 PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412, 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 379), at 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 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 434) 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:

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**, Cycle **11 SCALING**, and Cycle **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.

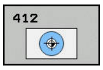
- Positioning of the touch points
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

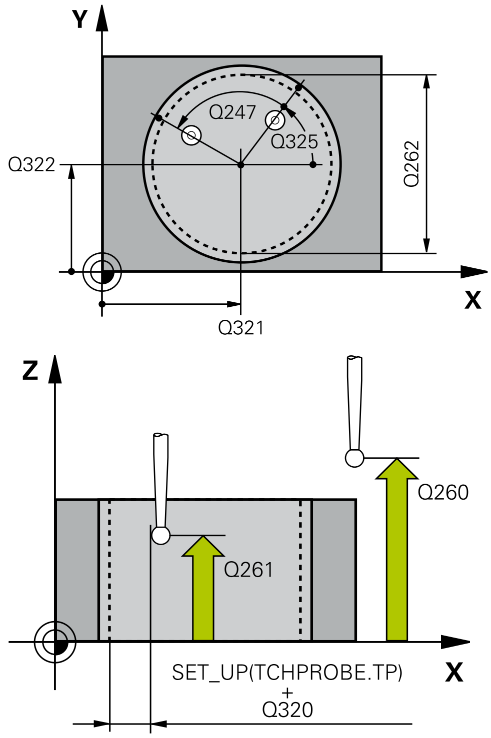
The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

Program the stepping angle to be less than 90°, input range: –120° to 120°

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the pocket in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. 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 principal 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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
Q305=12	;NUMBER IN TABLE

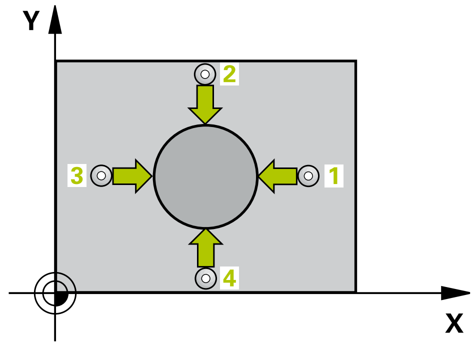
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset 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 principal 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 secondary 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 434)
  0: Write the determined 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).

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 principal 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 secondary 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

16.5 PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413, 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 379), at 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 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 434) 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:

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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **high** estimate for the nominal diameter of the stud.

- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

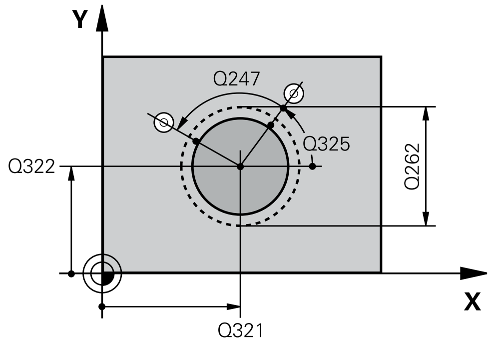
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

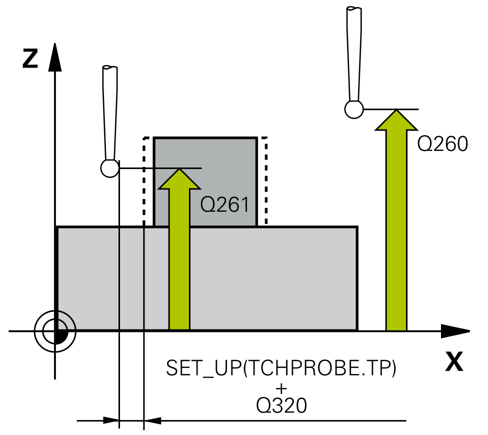
The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

Program the stepping angle to be less than 90°, input range: –120° to 120°

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the stud in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. 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 principal 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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

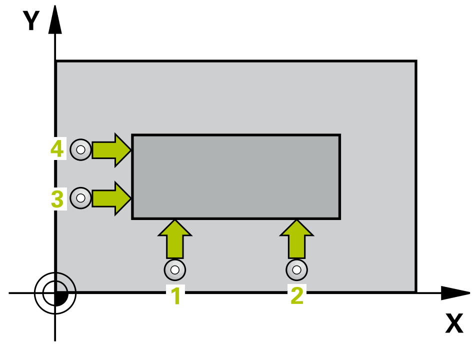
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes 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 principal 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 secondary 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 434)
  0: Write the determined 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 principal 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 secondary 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

16.6 PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414, option 17)

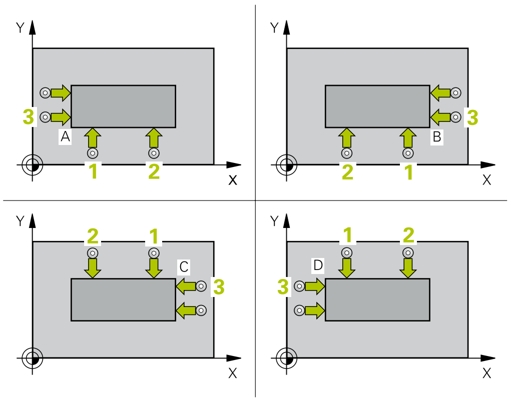
Cycle run

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 379), at 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 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 434) 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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

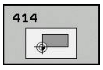
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control always measures the first line in the direction of the minor axis of the working plane.

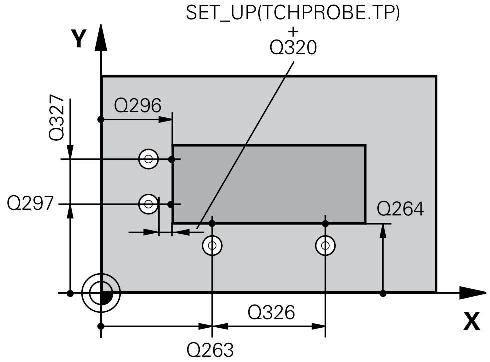
By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see figure at the right and table below).

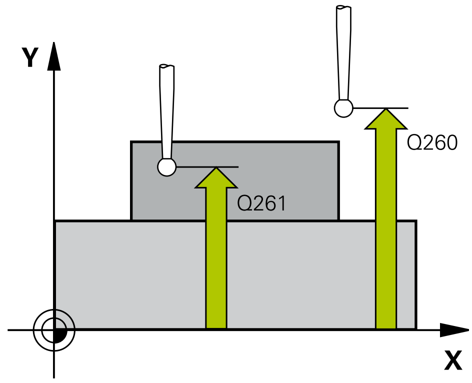
Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal 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 secondary 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 principal 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 principal 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 secondary 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 secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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 principal 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 secondary 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 434)
  0: Write the determined 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

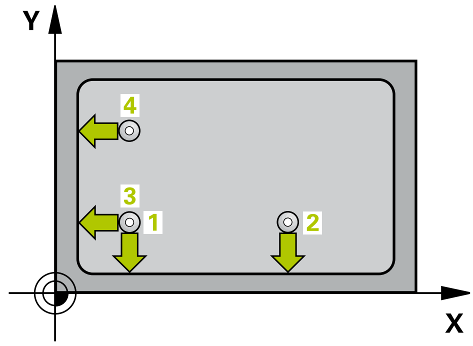
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal 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 secondary 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

16.7 PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415, 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 "Executing touch probe cycles", at touch point **1** (see figure at right). The control offsets the touch probe in the principal and secondary axes by the set-up clearance **Q320** + **SET_UP** + ball-tip radius (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 moves to the next touch point **2**; the control offsets the touch probe in the secondary axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the second probing operation
4. The control positions the touch probe at touch point **3** (same positioning logic as for the first touch point) and performs the probing operation there
5. Then the touch probe moves to touch point **4**. The control offsets the touch probe in the principal axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the fourth probing operation
6. Finally, the control retracts the touch probe to the clearance height. It processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 434) and saves the coordinates of the determined corner 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 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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

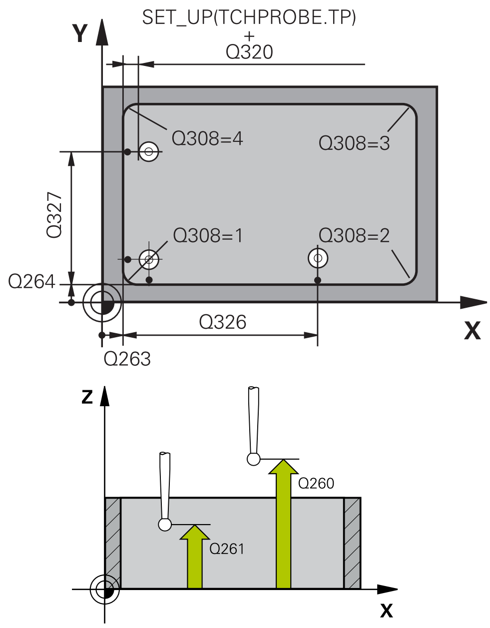
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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 corner in the principal axis of the working plane Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the corner in the secondary axis of the working plane Input range: –99999.9999 to 99999.9999
- **Q326 Spacing in 1st axis?** (incremental): Distance between the corner and second measuring point in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q327 Spacing in 2nd axis?** (incremental): Distance between the corner and fourth measuring point in the secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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

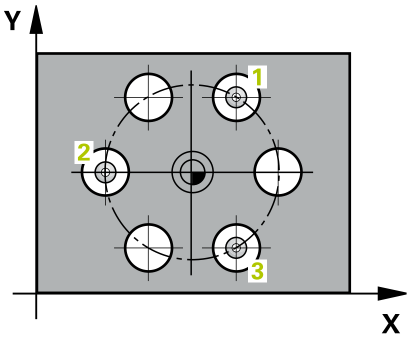
5 TCH PROBE 415 PRESET OUTS. CORNER	
Q263=+37	;1ST POINT 1ST AXIS
Q264=+7	;1ST POINT 2ND AXIS
Q326=50	;SPACING IN 1ST AXIS
Q327=45	;SPACING IN 2ND AXIS
Q308=+1	;CORNER
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q304=0	;BASIC ROTATION
Q305=7	;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 principal 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 secondary 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 434)
  0: Write the determined 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 principal 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 secondary 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

16.8 PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416, 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 379), 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 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 434) 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!

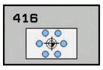
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

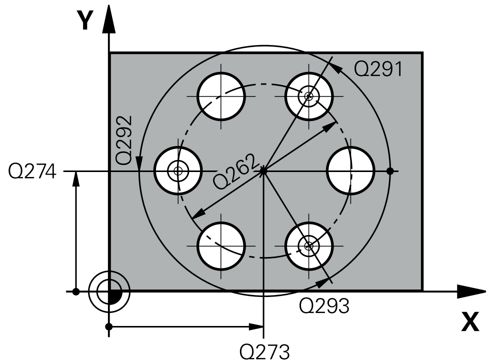
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

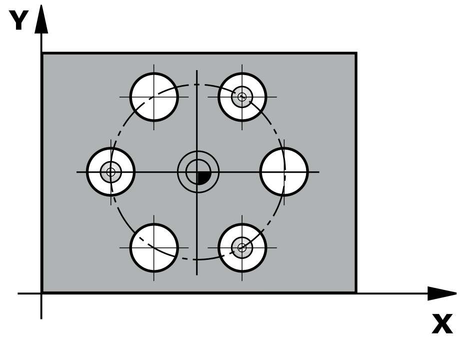
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the principal 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 secondary 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 principal axis at which the control should set the bolt-hole circle center. Default setting = 0. Input range: –99999.9999 to 99999.9999





Example

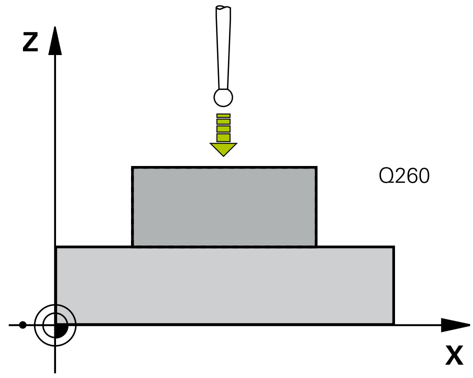
5 TCH PROBE 416 PRESET CIRCLE CENTER	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=90	;NOMINAL DIAMETER
Q291=+34	;ANGLE OF 1ST HOLE
Q292=+70	;ANGLE OF 2ND HOLE
Q293=+210	;ANGLE OF 3RD HOLE
Q261=-5	;MEASURING HEIGHT
Q260=+20	;CLEARANCE HEIGHT
Q305=12	;NUMBER IN TABLE
Q331=+0	;PRESET
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 secondary 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 434)
  0: Write the determined 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 principal 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 secondary 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 the touch 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

16.9 PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417, 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 379), at 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 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 434) and saves the actual value in the Q parameter 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.

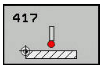
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

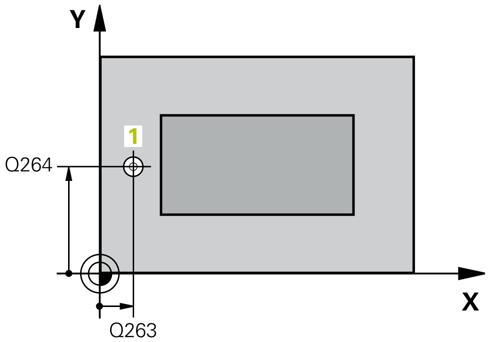
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

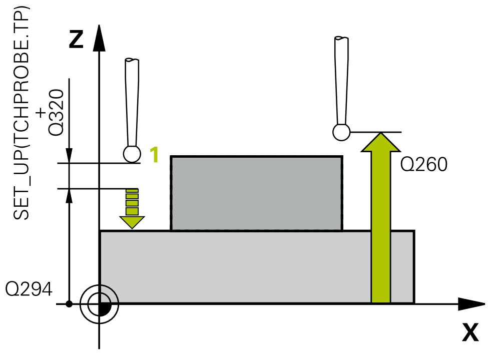
The control then sets the preset in this axis.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal 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 secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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 in the touch probe axis 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 434)
  0: Write the determined 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).





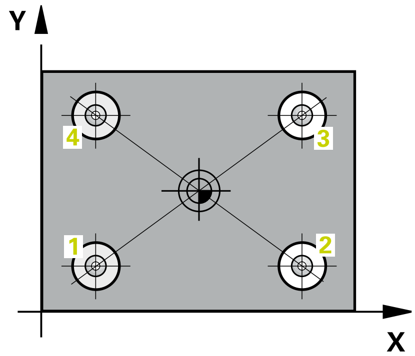
Example

```
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
```

16.10 PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418, 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 379), 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 434). 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!

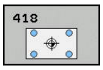
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

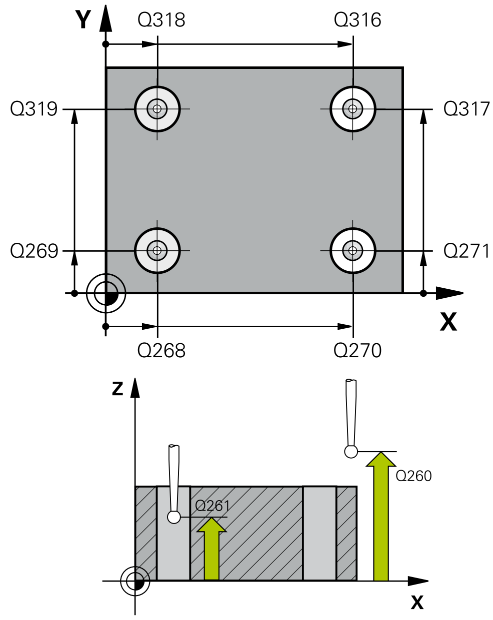
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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 principal 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 secondary 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 principal 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 secondary 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 principal 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 secondary 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 principal 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 secondary 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

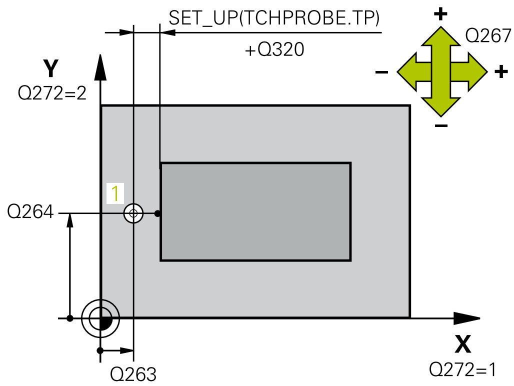
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates of the point of intersection of the connecting lines; input range: 0 to 9999.
  If **Q303 = 1**, the control will write the data to the preset table. If the active 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 principal 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 secondary 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 434)
  0: Write the determined 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 principal 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 secondary 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

16.11 PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419, 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 379), at the programmed touch point **1**. The control offsets the touch probe by the safety 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 434)

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

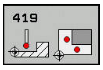
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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

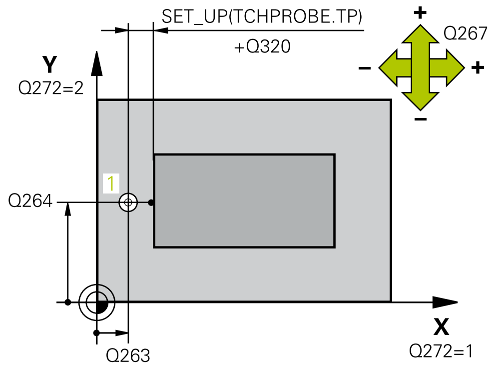


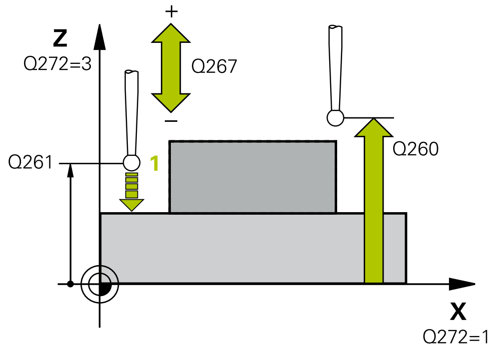
- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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: Principal axis = measuring axis
  2: Secondary 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

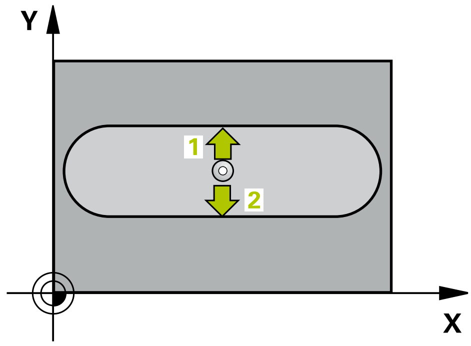
5 TCH PROBE 419 PRESET IN ONE AXIS	
Q263=+25	;1ST POINT 1ST AXIS
Q264=+25	;1ST POINT 2ND AXIS
Q261=+25	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+50	;CLEARANCE HEIGHT
Q272=+1	;MEASURING AXIS
Q267=+1	;TRAVERSE DIRECTION
Q305=0	;NUMBER IN TABLE
Q333=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER

- **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?** (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 434)
  0: Write the determined 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).

16.12 SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408, 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 379), at 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 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 434) 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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

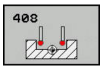
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **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.

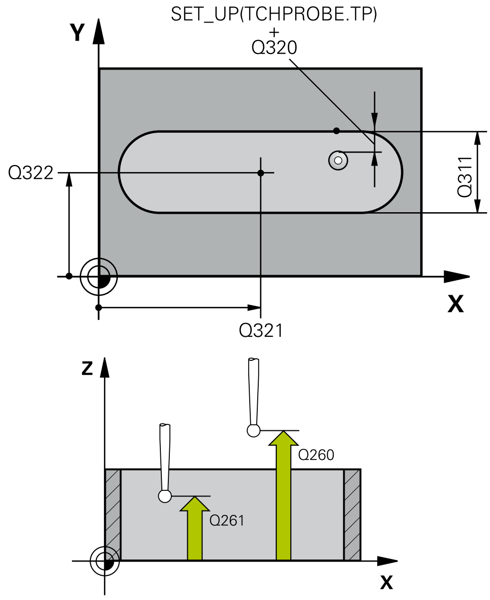
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the slot in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the slot in the secondary 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: Principal axis = measuring axis
  2: Secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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



Example

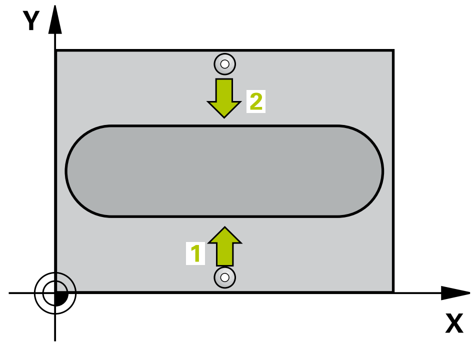
5 TCH PROBE 408 SLOT CENTER PRESET	
Q321=+50	**;CENTER IN 1ST AXIS**
Q322=+50	**;CENTER IN 2ND AXIS**
Q311=25	**;SLOT WIDTH**
Q272=1	**;MEASURING AXIS**
Q261=-5	**;MEASURING HEIGHT**
Q320=0	**;SET-UP CLEARANCE**
Q260=+20	**;CLEARANCE HEIGHT**
Q301=0	**;MOVE TO CLEARANCE**
Q305=10	**;NUMBER IN TABLE**
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 principal 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 secondary 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

16.13 RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409, 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 379), at 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 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 434) 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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

NOTICE

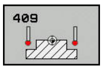
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

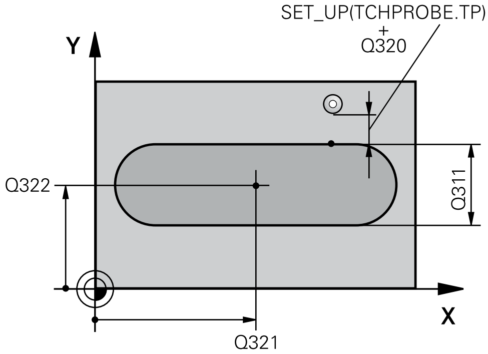
- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

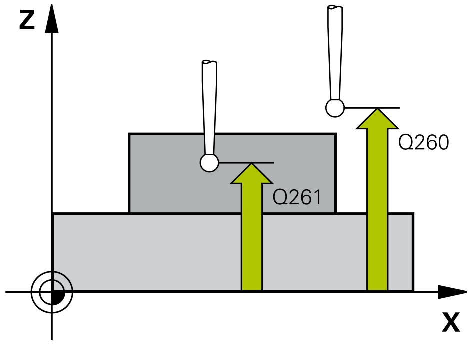
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the ridge in the principal axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the ridge in the secondary 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: Principal axis = measuring axis
  2: Secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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.





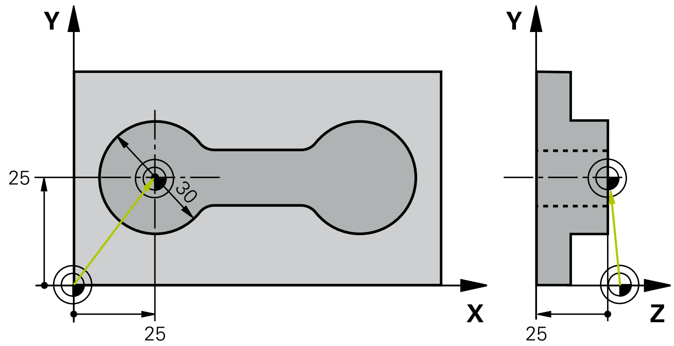
Example

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 principal 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 secondary 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

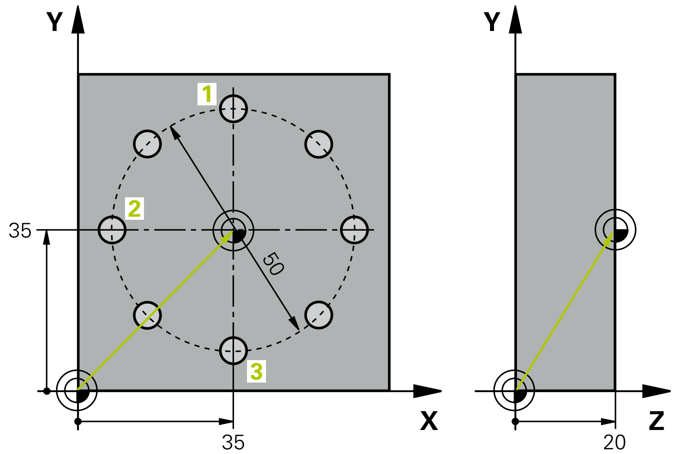
16.14 Example: Presetting in center of a circular segment and on top surface of workpiece



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	

16.15 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		

17

Touch Probe Cycles: Automatic Workpiece Inspection

17.1 Fundamentals

Overview

NOTICE

Danger of collision!

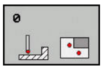
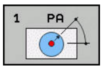
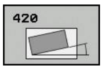
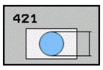
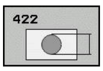
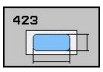
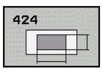
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

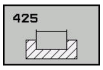
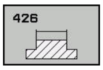
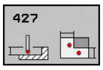
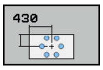
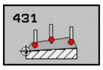
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

The control must be specifically 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	496
	1 POLAR PRESET Measuring a point, probing direction defined by an angle	497
	420 MEASURE ANGLE Measuring an angle in the working plane	498
	421 MEASURE HOLE Measuring the position and diameter of a hole	501
	422 MEASURE CIRCLE OUTSIDE Measuring the position and diameter of a circular stud	505
	423 MEASURE RECTANGLE INSIDE Measuring the position, length, and width of a rectangular pocket	509
	424 MEASURE RECTANGLE OUTSIDE Measuring the position, length, and width of a rectangular stud	513

Soft key	Cycle	Page
	425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width	516
	426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge	519
	427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis	522
	430 MEAS. BOLT HOLE (2nd soft-key row) Measuring the position and diameter of a bolt hole circle	525
	431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane	528

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.

> 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

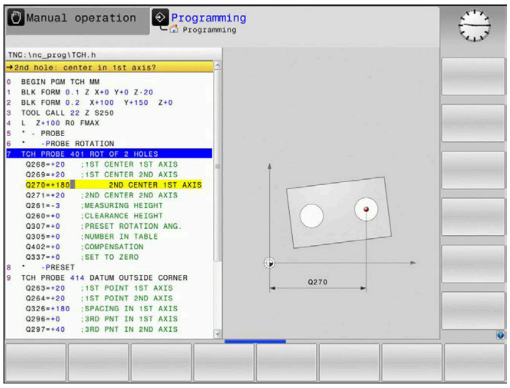
Nominal values:	
Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000
Given limit values:	
Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000
Minimum limit for center in minor 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 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 due to the deviations from the nominal value (values in **Q16x**)
- the deviations from the nominal value (values in **Q16x**) are greater than the tool breakage tolerance.

Tool compensation

> This function only works
> - 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).

If you want to compensate the values for an indexed tool with a tool name, program the following:

- **QS0** = "TOOL NAME"
- **FN18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0**= **Q0** +0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool

Tool breakage monitoring

This function only works

- 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

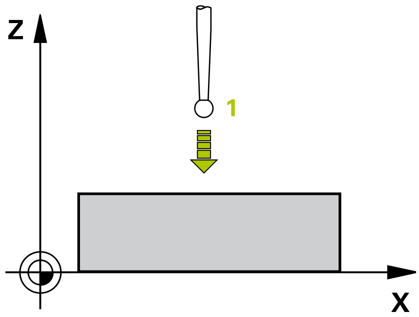
The control will output an error message and stop the program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time, the tool will be deactivated in the tool table (column TL = L).

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.

17.2 REFERENCE PLANE (Cycle 0, DIN/ISO: G55, 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

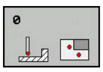
Danger of collision!

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

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters



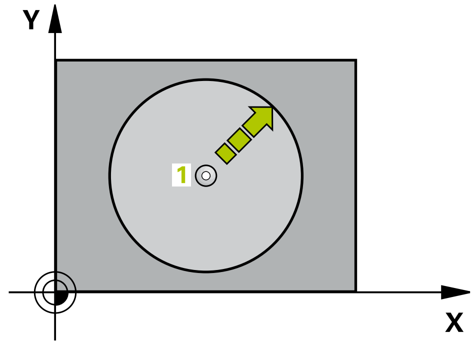
- **Parameter number for result?**: Enter the number of the Q parameter to which you want to assign the coordinate. Input range: 0 to 1999
- **Probing axis/probing direction?**: Select the probing axis with the axis key or the alphabetic keyboard, entering the algebraic sign for the probing direction. 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
```

17.3 POLAR PRESET (Cycle 1, 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). Use polar angles to define the probing direction in the cycle.
3 After the control has saved the position, the touch probe returns to the starting point. The control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**

Please note while programming:

NOTICE

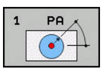
Danger of collision!

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

> This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
>
> 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.

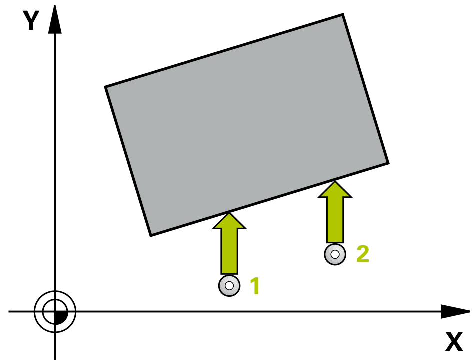
Example

```
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
```

17.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420, 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 379), at 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:

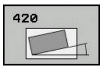
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

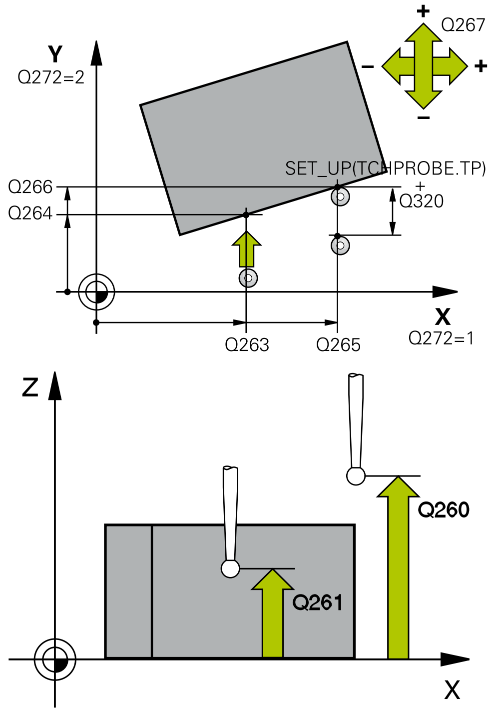
If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:

- 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 principal 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 secondary 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 principal 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 secondary 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: Principal axis = measuring axis
  2: Secondary 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



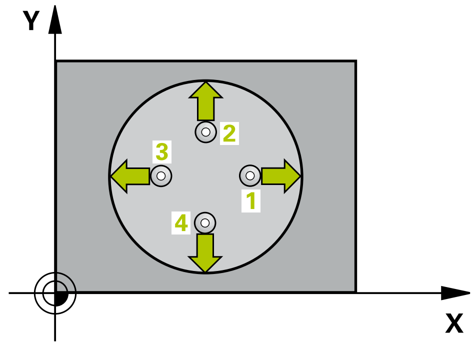
Example

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**)

17.5 MEASURE HOLE (Circle 421, DIN/ISO: G421, 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 379), at 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:

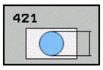
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

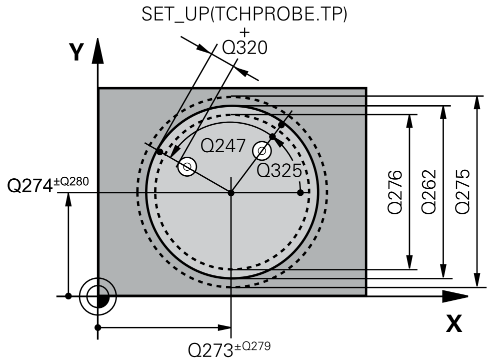
The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.

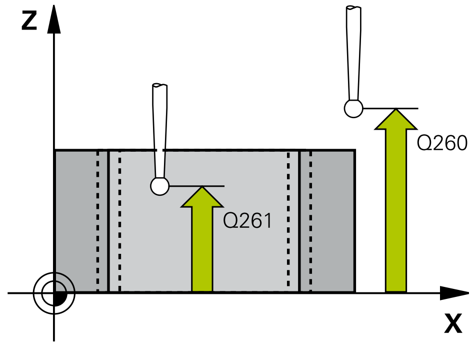
Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the hole in the principal 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 secondary 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 principal 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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=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
Q498=0	;REVERSE TOOL
Q531=0	;ANGLE OF INCIDENCE

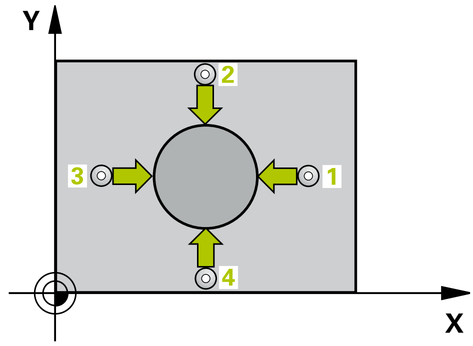
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q275 Maximum limit of size for hole?**: Maximum permissible diameter 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 principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary 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

- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 494). 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.

17.6 MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422, 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 379), at 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:

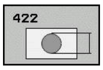
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

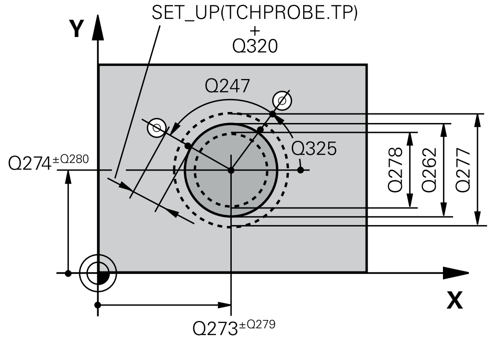
The smaller the stepping angle, the less accurately the control can calculate the dimensions of the stud. Minimum input value: 5°.

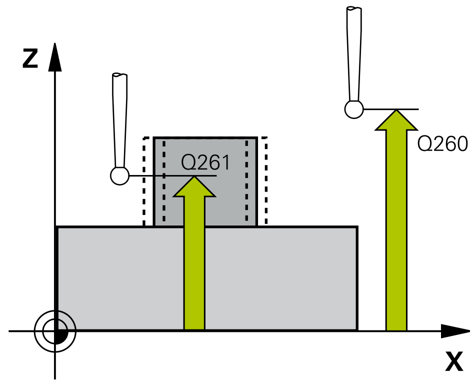
Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the principal 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 secondary 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 principal 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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 principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary 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 494). 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

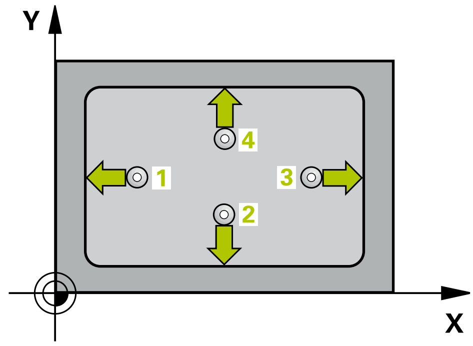
Q279=0.05	;TOLERANCE 1ST CENTER
Q280=0.05	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE
Q498=0	;REVERSE TOOL
Q531=0	;ANGLE OF INCIDENCE

- **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.

17.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423, 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 379), at 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

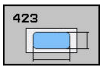
Please note while programming:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

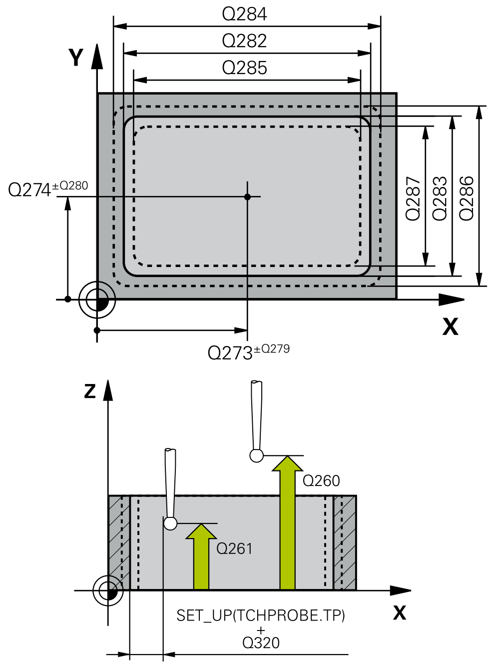
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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 principal 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 secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q282 1st side length (nominal value)?**: Pocket length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q283 2nd side length (nominal value)?**: Pocket length, parallel to the secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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
- **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 principal axis of the working plane. Input range: 0 to 99999.9999



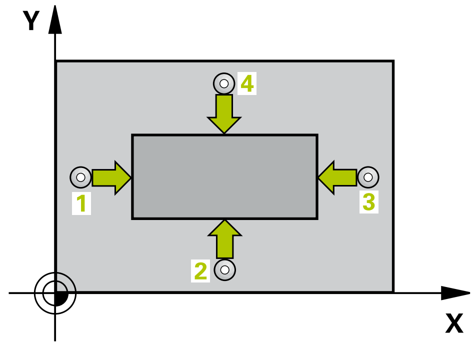
Example

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
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary 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 494). 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

17.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424, 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 379), 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:

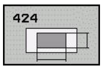
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

Please note while programming:

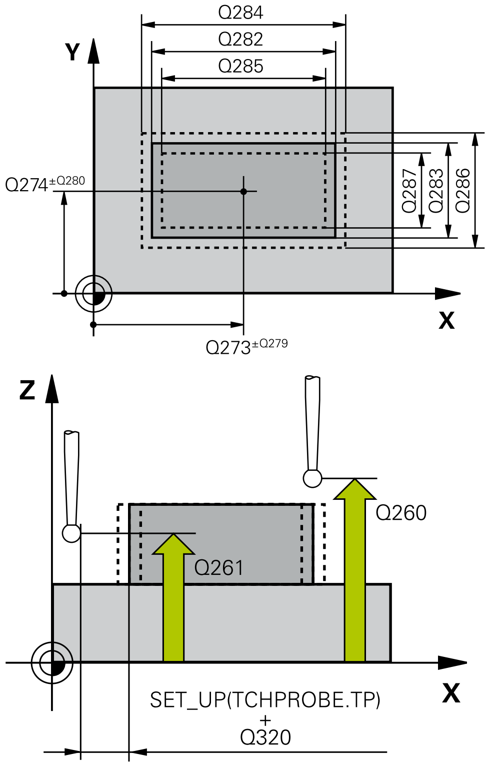
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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): Center of the stud in the principal 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 secondary axis of the working plane. Input range: –99999.9999 to 99999.9999
- **Q282 1st side length (nominal value)?**: Stud length, parallel to the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q283 2nd side length (nominal value)?**: Stud length, parallel to the secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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

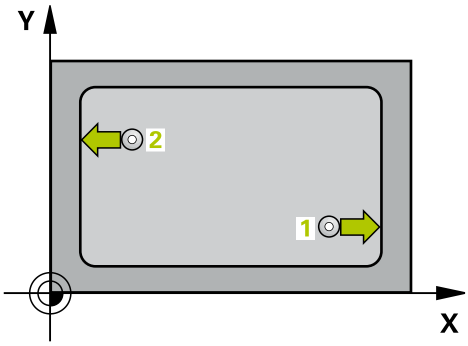
5 TCH PROBE 424 MEAS. RECTAN. OUTS.	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;2ND CENTER 2ND AXIS
Q282=75	;FIRST SIDE LENGTH
Q283=35	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q284=75.1	;MAX. LIMIT 1ST SIDE
Q285=74.9	;MIN. LIMIT 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 principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary 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 494). 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

17.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425, 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 "Executing touch probe cycles", at 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:

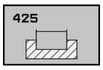
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Please note while programming:

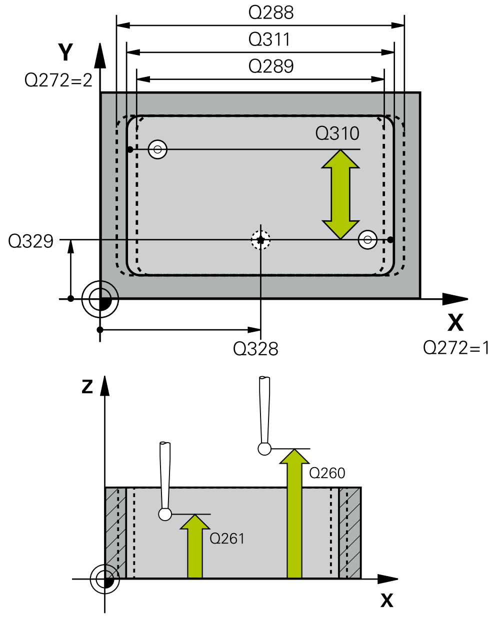
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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 principal 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 secondary 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: Principal axis = measuring axis
  2: Secondary 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: By default, the control will save the **log file named TCHPR425.TXT** 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

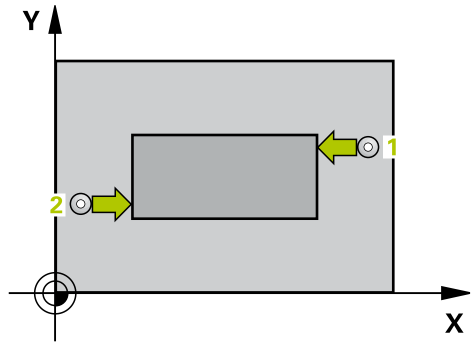
5 TCH PROBE 425 MEASURE INSIDE WIDTH	
Q328=+75	;STARTNG PNT 1ST AXIS
Q329=-12.5	;STARTNG PNT 2ND AXIS
Q310=+0	;OFFS. 2ND MEASUREMNT
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+10	;CLEARANCE HEIGHT
Q311=25	;NOMINAL LENGTH
Q288=25.05	;MAXIMUM LIMIT
Q289=25	;MINIMUM LIMIT
Q281=1	;MEASURING LOG
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 494). 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 the touch 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

17.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426, 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 379), 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:

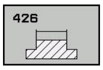
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Please note while programming:

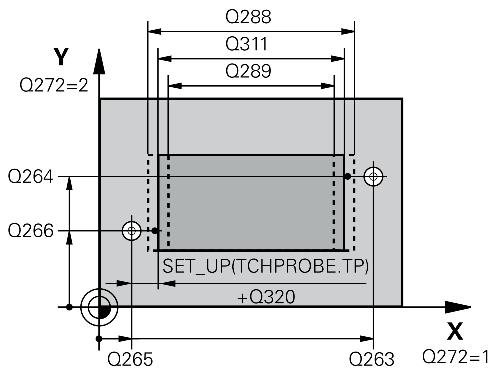
This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

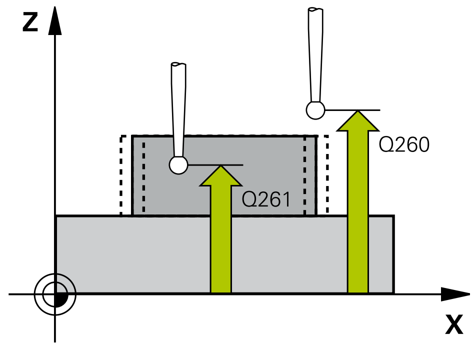
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal 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 secondary 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 principal 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 secondary 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: Principal axis = measuring axis
  2: Secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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**.





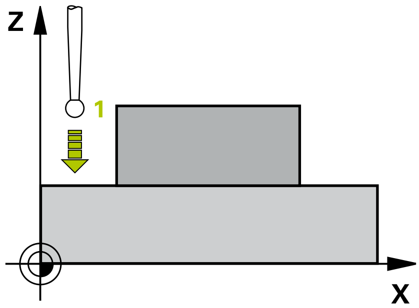
Example

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 494). 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.

17.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427, option 17)

Cycle run



The touch probe cycle 427 measures a coordinate in a selectable axis and saves the value in a Q parameter. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Using Touch Probe Cycles", at 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:

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

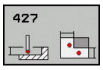
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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**).

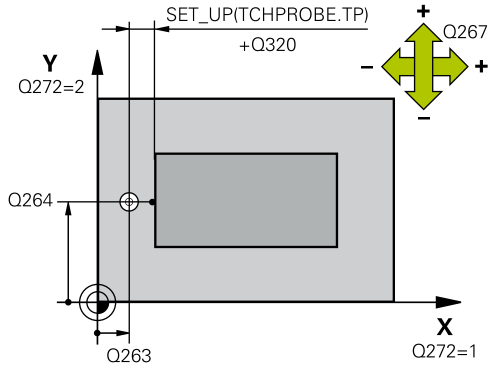
If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.

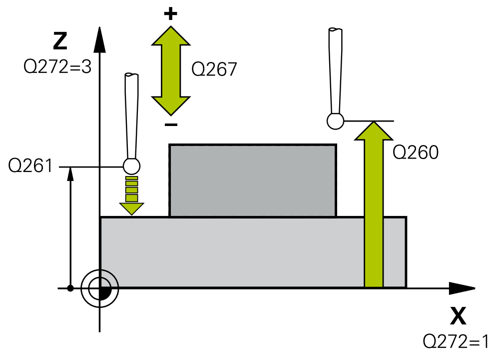
Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal 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 secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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: Principal axis = measuring axis
  2: Secondary 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
- **Q281Measuring 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





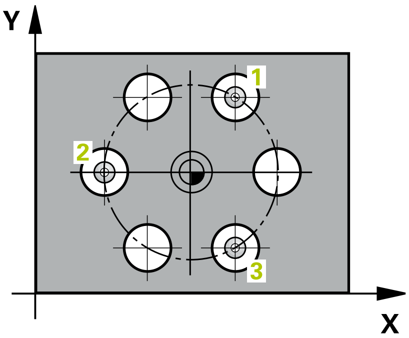
Example

```
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 494). 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.

17.12 MEAS. BOLT HOLE (Cycle 430, DIN/ISO: G430, 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.

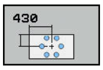
1. The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 379), to the programmed center point of the first hole **1**.
2. Then the probe moves to the entered measuring height and probes four points 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

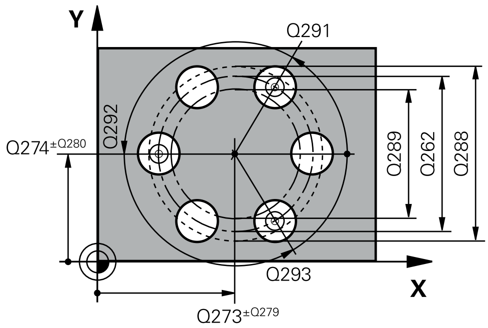
Please note while programming:

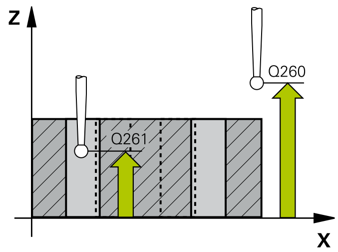
> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
>
> Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> Cycle 430 only monitors for tool breakage; there is no automatic tool compensation.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the principal 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 secondary 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
```

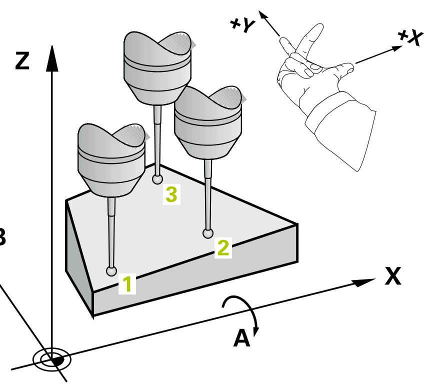
- **Q289 Minimum limit of size?**: Minimum permissible diameter of bolt hole circle. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary 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 494). 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

17.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431, 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 379), at 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 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:

NOTICE

Danger of collision!

If you save the values in the preset table and then tilt the tool by programming **PLANE SPATIAL** with **SPA**=0; **SPB**=0; **SPC**=0, there are multiple solutions in which the tilting axes are at 0.

- Make sure to program **SYM** (**SEQ**) + or **SYM** (**SEQ**) -

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

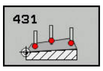
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control can only calculate the angular values if the three measuring points are not positioned on a straight line.

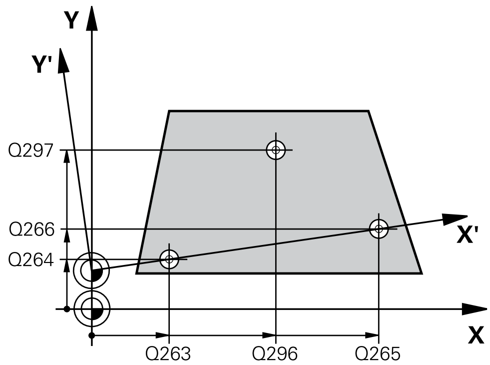
The spatial angles that are needed for the **Tilt working plane** function are saved in parameters **Q170** to **Q172**. With the first two measuring points you also specify the direction of the principal axis when tilting the working plane.

The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

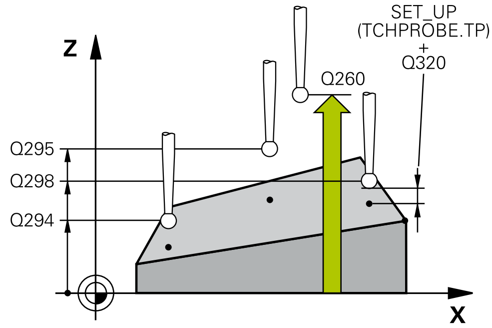
Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal 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 secondary 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 principal 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 secondary 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 principal 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 secondary 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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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

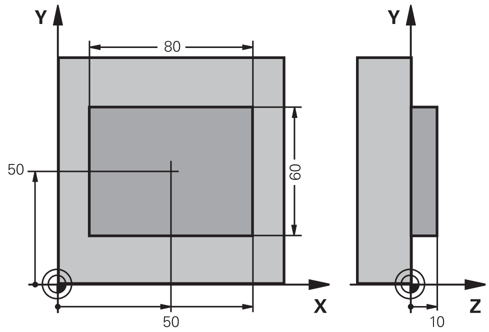
5 TCH PROBE 431 MEASURE PLANE
Q263=+20 ;1ST POINT 1ST AXIS
Q264=+20 ;1ST POINT 2ND AXIS
Q294=-10 ;1ST POINT 3RD AXIS
Q265=+50 ;2ND PNT IN 1ST AXIS
Q266=+80 ;2ND PNT IN 2ND AXIS
Q295=+0 ;2ND PNT IN 3RD AXIS
Q296=+90 ;3RD PNT IN 1ST AXIS
Q297=+35 ;THIRD POINT 2ND AXIS
Q298=+12 ;3RD PNT IN 3RD AXIS
Q320=0 ;SET-UP CLEARANCE
Q260=+5 ;CLEARANCE HEIGHT
Q281=1 ;MEASURING LOG

17.14 Programming Examples

Example: Measuring and reworking a rectangular stud

Program run

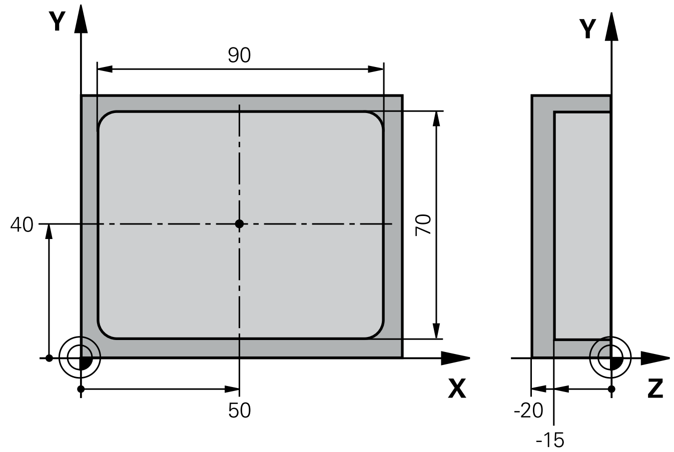
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



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 MILLING	
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		

18

Touch Probe Cycles: Special Functions

18.1 Fundamentals

Overview

NOTICE

Danger of collision!

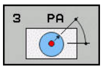
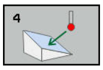
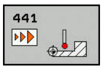
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**, Cycle **11 SCALING**, and Cycle **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	537
	4 **MEASURING IN 3-D** Measuring any position	539
	441 **FAST PROBING** Measuring cycle for defining various touch probe parameters	541

18.2 MEASURING (Cycle 3, 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.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other measuring cycles, do not apply in Touch Probe Cycle 3.

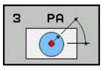
Remember that the control always writes to four successive Q parameters.

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. Save the measuring result in 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
```

18.3 MEASURING IN 3-D (Cycle 4, option 17)

Cycle run

> 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:

NOTICE

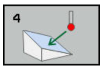
Danger of collision!

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.

> This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
>
> The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.
>
> 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.

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
```

18.4 FAST PROBING (Cycle 441, DIN/ISO: G441, 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:

Your machine tool builder may have limited the feed rate additionally. You can set the maximum absolute feed rate in the **maxTouchFeed** machine parameter (no. 122602).

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

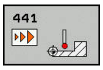
The purpose of Cycle 441 is to set parameters for probing cycles. In this cycle, no machine movements will be performed.

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.

Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397**=1.

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 with the feed rate potentiometer only, even with **Q397**=1. Your machine tool builder may have limited the feed rate additionally. You can set the maximum absolute feed rate in the **maxTouchFeed** machine parameter (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

18.5 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.

> Always calibrate a touch probe in the following cases:
> - Commissioning
> - Broken stylus
> - Stylus replacement
> - Change in the probe feed rate
> - Irregularities caused, for example, due to heating up of the machine
> - 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 are immediately effective. It is not necessary to repeat the tool call.

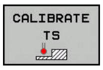
During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The control provides calibration cycles for calibrating the length and the radius:

Proceed as follows:

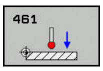
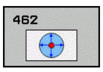
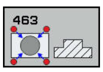
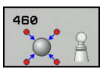
TOUCH PROBE

- Press the **TOUCH PROBE** key.



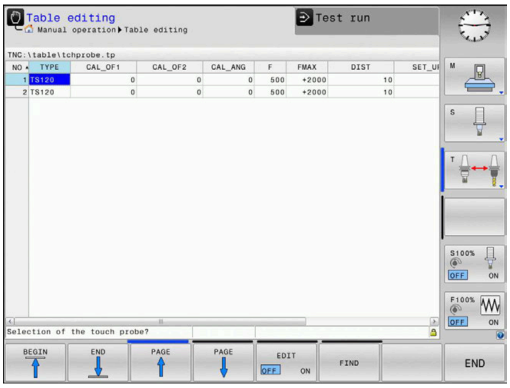
- Press the **CALIBRATE TS** soft key
- Select the calibration cycle

Calibration cycles provided by the control

Soft key	Function	Page
	Calibrating the length	545
	Measure the radius and the center offset using a calibration ring	547
	Measuring the radius and the center offset using a stud or a calibration pin	550
	Measure the radius and the center offset using a calibration sphere	553

18.6 Displaying calibration values

The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (principal axis) and **CAL_OF2** (secondary axis). 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:*.

Ensure that the tool number in the tool table and the touch-probe number of the touch-probe table match. This is regardless of whether you want to use a touch-probe cycle in automatic mode or in **Manual operation** mode.

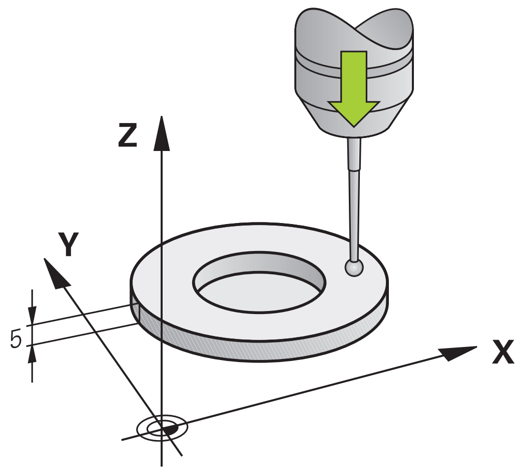
For more information, see Chapter Touch-probe table

18.7 TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461, 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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

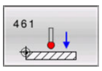
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

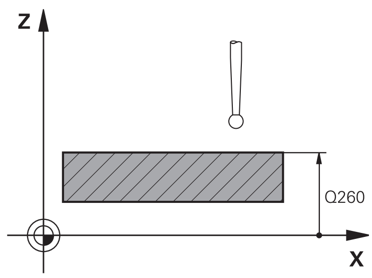
The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

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



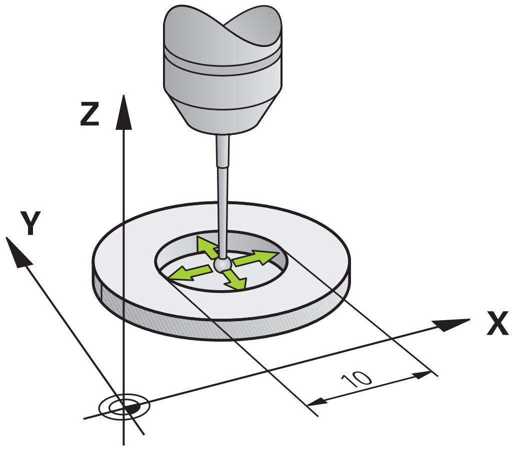
Example

```
5 TCH PROBE 461  TS CALIBRATION OF
   TOOL LENGTH
   Q434=+5      ;PRESET
```

18.8 CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462, option 17)

Cycle run

Refer to your machine manual!



Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows 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 routines. 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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

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 property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

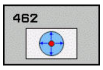
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

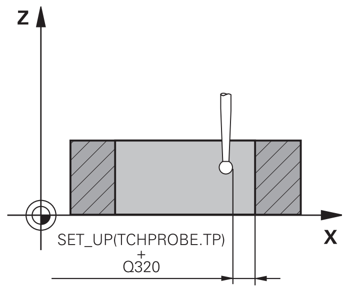
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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.



- **Q407 RING RADIUS** Enter the radius of the calibration ring. Input range: 0 to 9.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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 principal 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

18.9 TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463, 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 routines. 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**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand

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.

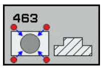
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

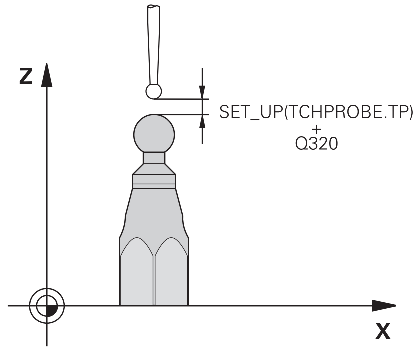
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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.



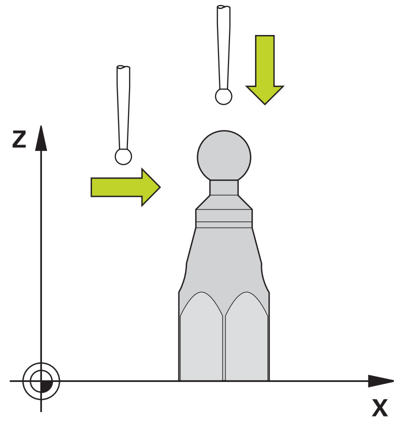
- **Q407 Radius of calibr. stud?**: Diameter of the calibration ring or pin. Input range: 0 to 99.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **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 principal 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

18.10 CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460, 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.

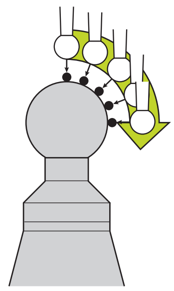
With Cycle 460 you can calibrate a triggering 3-D touch probe automatically using an exact calibration sphere.

It is also possible to capture 3-D calibration data. 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 are stored under TNC:\system\3D-ToolComp*. The DR2TABLE column of the tool table references the 3DTC table. The 3-D calibration data are 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 first movement is in the plane, depending on the reference angle (**Q380**).
4 The control then positions the touch probe in touch-probe axis.
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
6 Once the equator has been determined, the 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 first movement is in the plane, depending on the reference angle (**Q380**).
4 The control then positions the touch probe in touch-probe axis.
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
6 Once the equator has been determined, the 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**, Cycle **11 SCALING**, and Cycle **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.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (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.

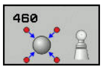
If you program **Q455**=0, the control will not perform a 3-D calibration.

If you program **Q455**=1 to 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 program **Q455**=1 to 30, a table will be stored under TNC:\system\3D-ToolComp*.

If there is already a reference to a calibration table (entry in DR2TABLE), this table will be overwritten.

If there is no reference to a calibration table (entry in DR2TABLE), then, in dependence of the tool number, a reference and the associated table will be created.



- **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 the touch 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 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

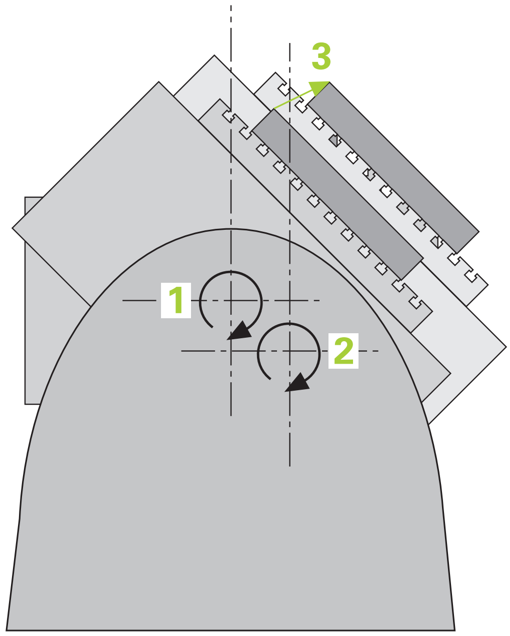
```
5 TCH PROBE 460  CALIBRATION OF TS
   ON A SPHERE
   Q407=12.5  ;SPHERE RADIUS
   Q320=0     ;SET-UP CLEARANCE
   Q301=1     ;MOVE TO CLEARANCE
   Q423=4     ;NO. OF PROBE POINTS
   Q380=+0    ;REFERENCE ANGLE
   Q433=0     ;CALIBRATE LENGTH
   Q434=-2.5  ;PRESET
   Q455=15    ;NO. POINTS 3-D CAL.
```

19

Touch Probe Cycles: Automatic Kinematics Measurement

19.1 Kinematics measurement with TS touch probes (option 48)

Fundamentals



Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

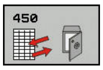
Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure at right) and the kinematic conditions actually existing on the machine (see **2** in the figure at right). When the rotary axes are positioned, these deviations 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	562
	451 **MEASURE KINEMATICS** Automatic checking or optimizing of machine kinematics	565
	452 **PRESET COMPENSATION** Automatic checking or optimizing of machine kinematics	580

19.2 Prerequisites

Refer to your machine manual!

Advanced Function Set 1 (option 8) must have been enabled.

Option 17 must have been enabled.

Option 48 must have been enabled.

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

The following are prerequisites for using the KinematicsOpt option:

- 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
- The kinematics description of the machine must be complete and correct, and the transformation dimensions must have been entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine 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 tool builder and is used to position the rotary axes.

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.

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**, Cycle **11 SCALING**, and Cycle **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.

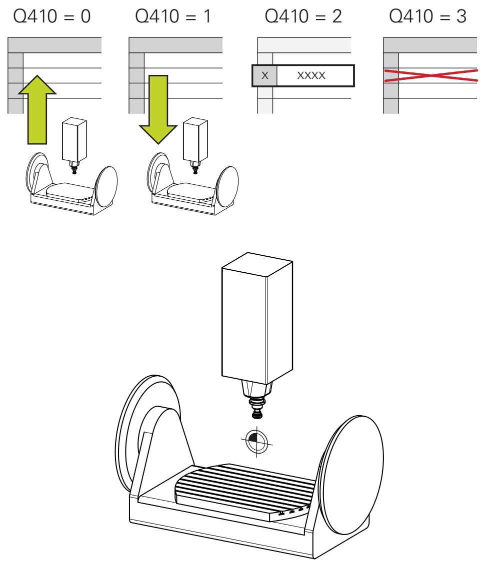
If an M function has been defined in the optional **mStrobeRotAxPos** machine parameter (no. 204803), you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for 450).

If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

19.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48)

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:

Only save and restore data with Cycle 450, while no tool carrier kinematics configuration that includes transformations is active.

This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.

Always save the active kinematics configuration before running a kinematics optimization. Advantage:

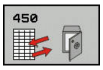
- You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).

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. So redefine the preset, if required.

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.

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
```

Log 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 are removed, too. If the data in the file are changed manually, the data records may become corrupted so that they are unusable.

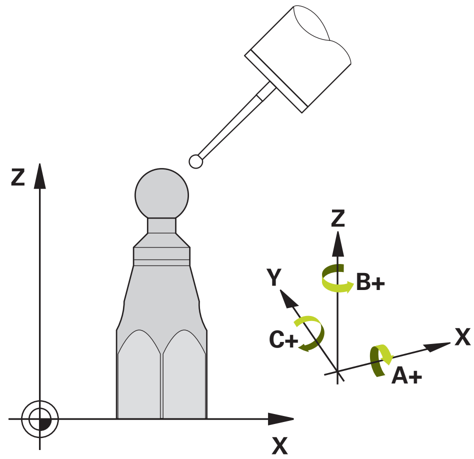
> If the **TNC:\table\DATA450.KD** file does not exist, it is generated automatically when Cycle 450 is executed.
>
> Make sure that you delete empty files with the name **TNC:\table\DATA450.KD**, if any, before starting Cycle 450. If there is an empty storage table (**TNC:\table\DATA450.KD**) that does 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).

19.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)

Cycle run

Refer to your machine manual!

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.



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 Operation mode, set the preset to the center of the sphere or, if you defined **Q431**=1 or **Q431**=3: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select the Program Run operating mode and start the calibration program.
4 The control automatically measures all rotary axes successively in the resolution you defined.
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 option 2 is not available.

The measured positions are calculated from the start angle, end angle, and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = –30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = (**Q412** – **Q411**) / (**Q414** – 1)

Calculated stepping angle = (90° - (-30°)) / (4 – 1) = 120 / 3 = 40°

Measuring position 1 = **Q411** + 0 * stepping angle = –30° –> –30°

Measuring position 2 = **Q411** + 1 * stepping angle = +10° –> 9°

Measuring position 3 = **Q411** + 2 * stepping angle = +50° –> 51°

Measuring position 4 = **Q411** + 3 * stepping angle = +90° –> 90°

Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1 or 2), 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.

> 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 performed with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432**, you can activate backlash measurement. Enter an angle that the control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.

> Backlash measurement is not possible if an M function for positioning the rotary axes is set in the optional **mStrobeRotAxPos** machine parameter (no. 204803) or if the axis is a Hirth axis.

> 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 "Log function", Page 579).

Please note while programming:

If the optional **mStrobeRotAxPos** machine parameter (no. 204803) is not equal to –1 (M function positions the rotary axes), only start a measurement if all rotary axes are at 0°.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional **maxDevCalBall** machine parameter (no. 204802), the control displays an error message and ends the measurement.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.

As with Cycles 451 and 452, Cycle 453 ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.

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.

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 adapt 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.

Angle compensation is only possible with option 52 **KinematicsComp**.

If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table. Input range: 0 to 99999.9999Alternatively **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. Fewer 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 pre-position 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): Pre-position 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

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 are adjusted automatically.

Position and Angle Optimization mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- First the control tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

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

Log function

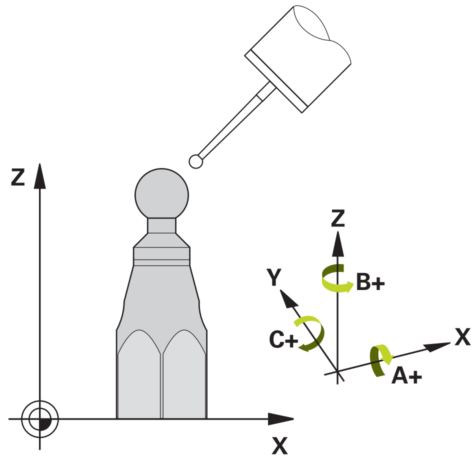
After running Cycle 451, the control will create a log **(TCHPR451.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- 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)

19.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option 48)

Cycle run

	Refer to your machine manual!



Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)", Page 565). 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:

If the kinematics data determined exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional **maxDevCalBall** machine parameter (no. 204802), the control displays an error message and ends the measurement.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.

As with Cycles 451 and 452, Cycle 453 ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.

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.

Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

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?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **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. Fewer 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, the adjustment of a fork head with A and C axes is described. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- Insert the interchangeable head that will be used as a reference head.
- Clamp the calibration sphere
- Insert the touch probe
- Use Cycle 451 to measure the complete kinematics, including the reference head.
- Define the preset (using **Q431** = 2 or 3 in Cycle 451) after measuring the reference head

Measuring a reference head

```
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
- Define the preset (using **Q432** = 2 or 3 in Cycle 451) after measuring the kinematics
- Then set the presets on your workpiece and start the machining process.

Reference measurement for drift compensation

```
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
```

Log 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 "Log function", Page 579)

20

Touch Probe Cycles: Automatic Tool Measurement

20.1 Fundamentals

Overview

Refer to your machine manual!

Some cycles and functions may not be provided on your machine.

Option 17 is required.

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

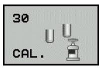
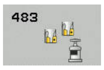
Operating notes

- When running touch probe cycles, Cycle **8 MIRROR IMAGE**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active
- HEIDENHAIN only guarantees the proper operation of the probing cycles if HEIDENHAIN touch probes are used.

In conjunction with the control's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius 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 the **Programming** mode of operation using the **TOUCH PROBE** key. The following cycles are available:

New format	Old format	Cycle	Page
		Calibrating the TT, Cycles 30 and 480	598
		Measuring the tool length, Cycles 31 and 481	600
		Measuring the tool radius, Cycles 32 and 482	604
		Measuring the tool length and radius, Cycles 33 and 483	608
		Calibrating the wireless TT 449, Cycle 484	612

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

The tool touch probe cycles 480, 481, 482, 483, and 484 can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).

Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200).

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.T tool table

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 cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from **offsetToolAxis** (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: 4 cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

20.2 Calibrating the TT (Cycle 30 or 480, DIN/ISO: G480, option 17)

Cycle run

Refer to your machine manual!

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 593). The calibration process runs automatically. The control also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylindrical pin. The resulting calibration values are stored in the control memory and are accounted for during subsequent tool measurement.

Calibration process:

1. Clamp the calibrating tool. The calibrating tool must be a precisely cylindrical part, for example a cylinder pin
2. Manually position the calibrating tool in the working plane via the center of the TT
3. Position the calibrating tool in the tool axis approx. 15 mm + safety clearance above the TT
4. The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
5. The calibration process along the tool axis starts
6. Calibration then follows in the working plane
7. The control positions the calibrating tool in the working plane at a position of TT radius + set-up clearance + 11 mm
8. Then the TNC moves the tool downwards along the tool axis and the calibration process starts
9. During probing, the control moves in a square pattern
10. The control saves the calibration values and considers them during subsequent tool measurement
11. The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Please note while programming:

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

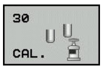
The functioning of the calibration cycle is dependent on machine parameter **CfgTTRoundStylus** (No. 114200). Refer to your machine manual.

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
```

20.3 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481, option 17)

Cycle run

Refer to your machine manual!

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:

NOTICE

Danger of collision!

To evaluate **Q199**, you need to set **stopOnCheck** (no. 122717) to **FALSE**. The NC program is not stopped when the breakage tolerance is exceeded. There is a danger of collision!

- Make sure that the NC program stops automatically if the breakage tolerance is exceeded.

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

You can run an individual tooth measurement of tools with **up to 20 teeth**.

Cycle parameters

- **Q340 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 breakage, 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.
- **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 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
- **Q341 Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- **For further information**, Page 603

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
```

Cycle 31 includes an additional parameter:

- **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**.

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
```

20.4 Measuring a tool radius (Cycle 32 or 482, ISO: G482, option 17)

Cycle run

Refer to your machine manual!

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 593). Select via input parameters by which of two methods the tool radius is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). The control probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Please note while programming:

NOTICE

Danger of collision!

To evaluate **Q199**, you need to set **stopOnCheck** (no. 122717) to **FALSE**. The NC program is not stopped when the breakage tolerance is exceeded. There is a danger of collision!

- Make sure that the NC program stops automatically if the breakage tolerance is exceeded.

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

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

- **Q340 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 breakage, 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.
- **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 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
- **Q341 Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- **For further information**, Page 607

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

Cycle 32 includes an additional parameter:

- **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**.

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
```

20.5 Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17)

Cycle run

Refer to your machine manual!

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 593). 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:

NOTICE

Danger of collision!

To evaluate **Q199**, you need to set **stopOnCheck** (no. 122717) to **FALSE**. The NC program is not stopped when the breakage tolerance is exceeded. There is a danger of collision!

- Make sure that the NC program stops automatically if the breakage tolerance is exceeded.

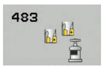
The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

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 parameter **CfgTT** (no. 122700). Refer to your machine manual.

Cycle parameters



- **Q340 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 breakage, 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, or DL, DR in the tool table.
- **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 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
- **Q341 Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- **For further information**, Page 611

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
```

Cycle 33 includes an additional parameter:

- **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**.

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
```

20.6 Calibrating the wireless TT 449 (Cycle 484, DIN/ISO: G484, option 17)

Fundamentals

With Cycle 484, you can calibrate your tool touch probe, e.g the wireless infrared TT 449 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle 484 you must move the tool over the TT

Cycle run

Refer to your machine manual!

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 and 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 run without stop. The calibration process starts from the current position of the tool.

Calibrating tool:

The calibrating tool must be a precisely cylindrical part, for example a cylindrical pin. Enter the exact length and radius of the calibrating tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Please note while programming:

NOTICE

Danger of collision!

To avoid collisions the tool must be pre-positioned before calling the cycle with **Q536**=1! The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

- Specify whether to stop before cycle start or run the cycle automatically without stopping.

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. If you use a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. Major inaccuracies may occur if you use a calibrating tool whose diameter is too small and/or that protrudes too far from the chuck.

Before calibrating the touch probe, you must enter the exact length and radius of the 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
```

21

Tables of Cycles

21.1 Table of cycles

Fixed cycles

Cycle number	Cycle name	DEF active	CALL active	Page
7	Datum shift	■		209
8	Mirroring	■		216
9	Dwell time	■		347
10	Rotation	■		218
11	Scaling factor	■		220
12	Program call	■		348
13	Oriented spindle stop	■		349
14	Contour definition	■		251
18	Thread cutting		■	369
19	Tilting the working plane	■		223
20	Contour data SL II	■		256
21	Pilot drilling SL II		■	258
22	Rough out SL II		■	260
23	Floor finishing SL II		■	265
24	Side finishing SL II		■	267
25	Contour train		■	272
26	Axis-specific scaling	■		221
27	Cylinder surface		■	313
28	Cylindrical surface slot		■	316
29	Cylinder surface ridge		■	320
32	Tolerance	■		350
39	Cylinder surface contour		■	323
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		■	119
207	Rigid tapping, new		■	122
208	Bore milling		■	100
209	Tapping with chip breaking		■	126
220	Polar pattern	■		236
221	Cartesian pattern	■		239

Cycle number	Cycle name	DEF active	CALL active	Page
224	Datamatrix code pattern	■		241
225	Engraving		■	354
232	Face milling		■	360
233	Face milling (selectable milling direction, consider the side walls)		■	196
238	Measure machine status	■		365
239	Ascertain the load	■		367
240	Centering		■	111
241	Single-lip deep-hole drilling		■	103
247	Presetting	■		230
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		■	133
263	Thread milling/countersinking		■	137
264	Thread drilling/milling		■	141
265	Helical thread drilling/milling		■	145
267	Outside thread milling		■	149
270	Contour train data		■	270
271	OCM contour data		■	297
272	OCM roughing		■	299
273	OCM finishing floor		■	302
274	OCM finishing side		■	304
275	Trochoidal slot		■	276
276	3-D contour train		■	282

Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	Reference plane	■		496
1	Polar preset	■		497
3	Measuring	■		537
4	Measuring in 3-D	■		539
30	Calibrating a TT	■		598
31	Measure/Inspect the tool length	■		600
32	Measure/Inspect the tool radius	■		604
33	Measure/Inspect the tool length and the tool radius	■		608
400	Basic rotation: Rot of 2 points	■		407
401	Basic rotation: Rot. of 2 holes	■		410
402	Basic rotation: Rot. of 2 studs	■		414
403	Compensate misalignment with rotary axis	■		419
404	Set basic rotation	■		428
405	Compensate misalignment with the C axis	■		424
408	Slot center preset (FCL 3 function)	■		477
409	Ridge center preset (FCL 3 function)	■		481
410	Preset inside rectangle	■		435
411	Preset outside rectangle	■		439
412	Preset from inside of circle (hole)	■		443
413	Preset from outside of circle (stud)	■		448
414	Preset from outside of corner	■		453
415	Preset from inside of corner	■		458
416	Preset circle center	■		463
417	Preset in TS axis	■		467
418	Preset from 4 holes	■		469
419	Preset in one selectable axis	■		474
420	Workpiece—measure angle	■		498
421	Measure circle outside (hole)	■		501
422	Measure circle inside (stud)	■		505
423	Workpiece—measure rectangle from inside	■		509
424	Workpiece—measure rectangle from outside	■		513
425	Measure inside width (slot)	■		516
426	Measure outside width (ridge)	■		519
427	Workpiece—measure in any selectable axis	■		522
430	Workpiece—measure bolt hole circle	■		525
431	Workpiece—measure plane	■		528

Cycle number	Cycle name	DEF active	CALL active	Page
441	Fast probing	■		541
450	KinematicsOpt: Save Kinematics (option)	■		562
451	KinematicsOpt: Measure Kinematics (option)	■		565
452	KinematicsOpt: Preset compensation	■		558
460	Calibrating the touch probe	■		553
461	Calibrate touch probe length	■		545
462	Calibrate touch probe inside radius	■		547
463	Calibrate touch probe outside radius	■		550
480	Calibrate TT	■		598
481	Measure/Inspect the tool length	■		600
482	Measure/Inspect the tool radius	■		604
483	Measure/Inspect the tool length and the tool radius	■		608
484	Calibrate TT	■		612
1410	Probing on edge	■		397
1411	Probing two circles	■		401
1420	Probing in plane	■		393

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.de

www.klartext-portal.com
The Information Site for HEIDENHAIN Controls

Klartext App
The Klartext on Your Mobile Device

Google Play Store

Apple App Store

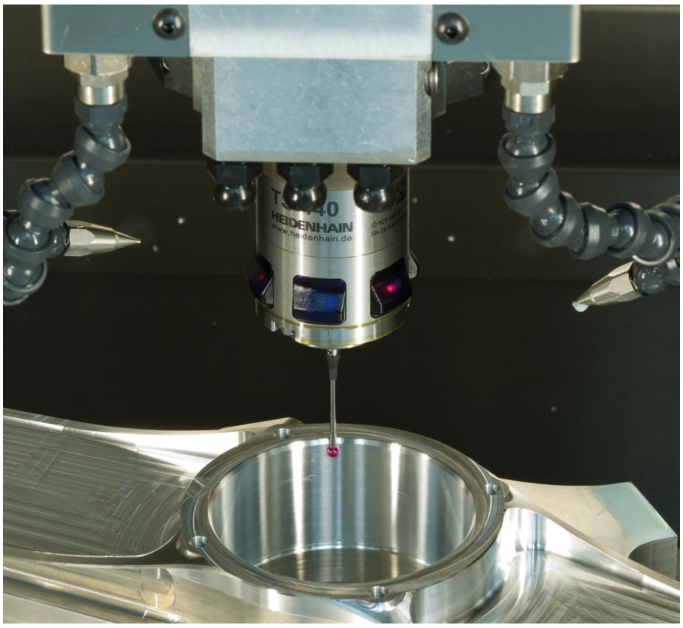
Touch probes from HEIDENHAIN

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	Infrared transmission
TS 642, TS 740	Infrared transmission

- Workpiece alignment
- Preset setting
- Workpiece measurement



Tool touch probes

TT 160	Signal transmission by cable
TT 460	Infrared transmission

- Tool measurement
- wear monitoring
- tool breakage detection

Original document
1096886-26 · Ver06 · SW07 · 10/2019 · H · Printed in Germany