HEIDENHAIN

TNC 640

User's Manual for Cycle Programming

NC Software
340590-09
340591-09
340595-09

English (en)
10/2018

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

Control model	NC software number
TNC 640	340590-09
TNC 640 E	340591-09
TNC 640 Programming station	340595-09

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

- Simultaneous linear movement in up to four axes

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

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

- Tool measurement with the TT

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

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

Operating instructions:

All control functions not related to the cycles are described in the TNC 640 User’s Manual. This manual is available from HEIDENHAIN upon request.

Conversational Programming User’s Manual ID: 892903-xx

ISO Programming User’s Manual ID: 892909-xx

User's Manual for Setup, Testing and Running NC programs ID: 1261174-xx

Software options

The TNC 640 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 (options 0 to 7)	
Additional axis	Additional control loops 1 to 8

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)

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

Display Step (option 23)	
Display step	**Input resolution:** ■ Linear axes down to 0.01 µm ■ Rotary axes to 0.00001°

Dynamic Collision Monitoring – DCM (option 40)	
Dynamic Collision Monitoring	■ The machine manufacturer defines objects to be monitored ■ Warning in Manual operation ■ Collision monitoring in the Test Run mode ■ Program interrupt in Automatic operation ■ Includes monitoring of 5-axis movements

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

Adaptive Feed Control – AFC (option 45)	
Adaptive Feed Control	**Milling:** ■ Recording the actual spindle power by means of a teach-in cut ■ Defining the limits of automatic feed rate control ■ Fully automatic feed control during program run **Turning (option 50):** ■ Cutting force monitoring during machining
KinematicsOpt (option 48)	
Optimizing the machine kinematics	■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics
Mill-Turning (option 50)	
Milling and turning modes	**Functions:** ■ Switching between Milling/Turning mode of operation ■ Constant surface speed ■ Tool-tip radius compensation ■ Turning cycles ■ Cycle 880: Gear hobbing (option 50 and option 131)
KinematicsComp (option 52)	
Three-dimensional compensation Export license required	Compensation of position and component errors
3D-ToolComp (option 92)	
3-D tool radius compensation depending on the tool’s contact angle Export license required	■ Compensate the deviation of the tool radius depending on the tool’s contact angle ■ Compensation values in a separate compensation value table ■ Prerequisite: Working with surface normal vectors (**LN** blocks)
Extended Tool Management (option 93)	
Extended tool management	Python-based
Advanced Spindle Interpolation (option 96)	
Interpolating spindle	**Interpolation turning:** ■ Cycle 291: Interpolation turning, coupling ■ Cycle 292: Interpolation turning, contour finishing
Spindle Synchronism (option 131)	
Spindle synchronization	■ Synchronization of milling spindle and turning spindle ■ Cycle 880: Gear hobbing (option 50 and option 131)
Remote Desktop Manager (option 133)	
Remote operation of external computer units	■ Windows on a separate computer unit ■ Incorporated in the control's interface

Synchronizing Functions (option 135)	
Synchronization functions	**Real Time Coupling – RTC:** Coupling of axes
Visual Setup Control – VSC (option 136)	
Camera-based monitoring of the setup situation	■ Record the setup situation with a HEIDENHAIN camera system ■ Visual comparison of planned and actual status in the workspace
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
Gear Cutting (option 157)	
Machining gear systems	■ Cycle 285: Define gear wheel ■ Cycle 286: Gear hobbing ■ Cycle 287: Gear skiving

Advanced Function Set Turning (option 158)	
Advanced turning functions	Cycle 883: Simultaneous turning

Feature Content Level (upgrade functions)

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

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

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

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

Intended place of operation

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

Legal information

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

- Programming operating mode
- MOD function
- **License Info** softkey

Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, which were not all available in some older software versions. Within a cycle, they are always provided at the end of the cycle definition. The section "New or changed cycle functions of software 34059x-09 " gives you an overview of the optional Q parameters that have been added in this software version. You can decide for yourself whether you would like to define optional Q parameters or delete them with the NO ENT key. You can also adopt the default value. If you have accidentally deleted an optional Q parameter or if you would like to extend cycles in your existing NC programs after a software update, you can add optional Q parameters in cycles where needed. The following steps describe how this is done.

To insert optional Q parameters in existing programs:

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

Compatibility

Most NC programs created with older HEIDENHAIN contouring controls (as of TNC 150 B) can be run with the new software version of the TNC 640. Even if new optional parameters ("Optional parameters") have been added to existing cycles, you will generally be able to run your NC programs as usual. This is achieved because the stored default value will be used. The other way round, if you want to run an NC program created with a new software version on an older control, you can delete the respective optional Q parameters from the cycle definition with the NO ENT key. In this way you can ensure that the NC program is be downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

New and changed cycle functions of software 34059x-08

- New Cycle 453 KINEMATICS GRID. This cycle makes it possible to probe a calibration sphere in multiple tilting-axis positions predefined by the machine tool builder. The measured deviations can be compensated via compensation tables. Options 48 KinematicsOpt and 52 KinematicsComp are required; the machine tool builder has to adapt the feature to the respective machine. see "KINEMATICS GRID (Cycle 453, ISO: G453, option)", Page 776
- New Cycle 441 FAST PROBING. With this cycle you can set various touch probe parameters (e.g. positioning feed rate) that are globally effective for all subsequently used touch probe cycles. see "FAST PROBING (Cycle 441, ISO G441)", Page 717
- New Cycle 276 3-D Contour Train see "THREE-D CONT. TRAIN (Cycle 276, ISO: G276)", Page 253
- Enhancement of the contour train: Cycle 25 with machining of residual material, the cycle was expanded with the following parameters: Q18, Q446, Q447, Q448 see "CONTOUR TRAIN (Cycle 25, ISO: G125)", Page 249
- Cycles 256 RECTANGULAR STUDand 257 CIRCULAR STUD were extended by the Q215, Q385, Q369, and Q386 parameters. see "RECTANGULAR STUD (Cycle 256, ISO: G256)", Page 189, see "CIRCULAR STUD (Cycle 257, ISO: G257)", Page 194
- The recessing cycles 860 to 862 and 870 to 872 were expanded by the input parameter Q211. In this parameter, a dwell time can be specified in revolutions of the workpiece spindle, which retards the retraction after the recessing on the floor. see "RECESSING CONTOUR RADIAL (Cycle 860, ISO: G860)", Page 482, see "RADIAL RECESSING (Cycle 861, ISO: G861)", Page 475, see "RADIAL RECESSING EXTENDED (Cycle 862, ISO: G862)", Page 478, see "AXIAL RECESSING (Cycle 870, ISO: G870)", Page 494, see "AXIAL RECESSING (Cycle 871, ISO: G871)", Page 486, see "AXIAL RECESSING EXTENDED (Cycle 872, ISO: G872)", Page 489
- Cycle 239 can ascertain the current load of the machine axes with the control function LAC. Cycle 239 can now also modify the maximum axis acceleration. Cycle 239 supports the determination of the load on synchronized axes. see "ASCERTAIN THE LOAD (Cycle 239, ISO: G239, software option 143)", Page 369
- Cycles 205 and 241: The feed rate behavior was modified. see "SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241, ISO: G241)", Page 111, see "UNIVERSAL PECKING (Cycle 205, ISO: G205)", Page 100
- Detail changes with Cycle 233: Monitors length of the cutting edge (LCUTS) with finishing operations, when roughing with the milling strategy between 0 to 3, the surface in the milling direction is increased by the value from Q357 (provided that no limitation is set in this direction) see "FACE MILLING (Cycle 233, ISO: G233)", Page 204
- CONTOUR DEF can be programmed in ISO format

- The technically obsolete cycles 1, 2, 3, 4, 5, 17, 212, 213, 214, 215, 210, 211, 230, 231 subordinated in "old cycles" can no longer be inserted via the editor. These cycles can however still be modified and executed.
- The tool touch probe cycles 480, 481, 482, 483, 484 can be hidden, see "Setting machine parameters", Page 786
- Cycle 225 Engraving can engrave the current counter reading with a new syntax see "Engraving the counter reading", Page 363
- New column SERIAL in the touch-probe table see "Touch probe data", Page 541

New or changed cycle functions of software 34059x-09

- New Cycle 285 DEFINE GEAR (software option 157), see "DEFINE GEAR (Cycle 285, ISO: G285, software option 157)", Page 376
- New Cycle 286 GEAR HOBBING (software option 157), see "GEAR HOBBING (Cycle 286, ISO: G286, software option 157)", Page 379
- New Cycle 287 GEAR SKIVING (software option 157), see "GEAR SKIVING (Cycle 287, ISO: G287, software option 157)", Page 385
- New Cycle 883 TURNING SIMULTANEOUS FINISHING(software options 50 and 158), see "TURNING SIMULTANEOUS FINISHING (Cycle 883, ISO: G883, software option #158) ", Page 511
- New Cycle 1410 PROBING ON EDGE (software option 17), see "PROBING ON EDGE (Cycle 1410, ISO: G1410)", Page 554
- New Cycle 1411 PROBING TWO CIRCLES (software option 17),see "PROBING TWO CIRCLES (Cycle 1411, ISO G1411)", Page 558
- New Cycle 1420 PROBING IN PLANE (software option 17), see "PROBING IN PLANE (Cycle 1420, ISO: G1420)", Page 550
- The simulation considers values sensed by a simulation probe. The simulation will be completed without error messages.
- In Cycle 24 SIDE FINISHING, a tangential helix will be used for rounding in the last infeed step, see "SIDE FINISHING (Cycle 24, ISO: G124)", Page 246
- Cycle 233 FACE MILLING, was extended by parameter Q367, SURFACE POSITION, see "FACE MILLING (Cycle 233, ISO: G233)", Page 204
- Cycle 257 CIRCULAR STUD, now uses Q207 FEED RATE FOR MILLNG for roughing, too, see "CIRCULAR STUD (Cycle 257, ISO: G257)", Page 194
- In Cycles 291 COUPLG.TURNG.INTERP. and 292 CONTOUR.TURNG.INTRP., the CfgGeoCycle configuration (no. 201000) is taken into account, see "COUPLING INTERPOLATION TURNING (Cycle 291, ISO: G291, software option 96)", Page 350 see "INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, ISO: G292, software option 96)", Page 338
- The touch probe cycles 408 to 419 consider chkTiltingAxes (no. 204600) for presetting, see "Touch Probe Cycles: Automatic Presetting", Page 589
- 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 589
- When pre-positioning the tool with Cycle 420 MEASURE ANGLE, the data both from the cycle and from the touch probe table will be taken into account, see "MEASURE ANGLE (Cycle 420, ISO: G420)", Page 656

- The help graphic in Cycle 444 PROBING IN 3-D for Q309 ERROR REACTION was changed; in addition, this Cycle now considers a TCPM, see "3D PROBING (Cycle 444)", Page 699
- In Cycle 444 PROBING IN 3-D, the positions of the rotary axes relative to the tilt angles are checked depending on the setting of the optional machine parameter, see "3D PROBING (Cycle 444)", Page 699
- Cycle 450 SAVE KINEMATICS, no longer writes identical values when restoring, see "SAVE KINEMATICS (Cycle 450, ISO: G450, option)", Page 748
- In Cycle 451 MEASURE KINEMATICS, the value 3 was added to cycle parameter Q406 MODE, see "MEASURE KINEMATICS (Cycle 451, ISO: G451, option)", Page 751
- In Cycles 451 MEASURE KINEMATICS and 453 KINEMATICS GRID, the radius of the calibration sphere is monitored in the second measurement only, see "MEASURE KINEMATICS (Cycle 451, ISO: G451, option)", Page 751 see "KINEMATICS GRID (Cycle 453, ISO: G453, option)", Page 776
- In Cycle 800 ADJUST XZ SYSTEM, the precision of the Q531 ANGLE OF INCIDENCE parameter was changed to 0.001°
- A REACTION column was added to the touch probe table, see "Touch-probe table", Page 540
- The CfgThreadSpindle machine parameter (no. 113600) is now available, see "TAPPING with a floating tap holder (Cycle 206, ISO: G206)", Page 125 , see "TAPPING without a floating tap holder (rigid tapping) GS (Cycle 207, ISO: G207)", Page 128, see "TAPPING WITH CHIP BREAKING (Cycle 209, ISO: G209)", Page 133 , see "THREAD CUTTING (Cycle 18, ISO: G18)", Page 371

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

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

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring and counterboring	80
DRILLING/ THREAD	Cycles for tapping, thread cutting and thread milling	124
POCKETS/ STUDS/ SLOTS	Cycles for milling pockets, studs and slots and for face milling	164
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	304
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	272
PATTERN	Cycles for producing point patterns, such as circular or linear hole patterns	218
TURNING	Cycles for turning and gear hobbing	402
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning, determin-ing the load, gear cycles	330

▷

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

Overview of touch probe cycles

TOUCH PROBE

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

Soft key	Cycle group	Page
ROTATION	Cycles for automatic measurement and compensation of workpiece misalignment	543
PRESET	Cycles for automatic workpiece presetting	590
MEASURING	Cycles for automatic workpiece inspection	648
SPECIAL CYCLES	Special cycles	694
CALIBRATE TS	Touch probe calibration	706
KINEMATICS	Cycles for automatic kinematics measurement	743
TT CYCLES	Cycles for automatic tool measurement (enabled by the machine manufacturer)	784
MONITORING WITH CAMERA	Cycles for VSC: camera-based monitoring of the setup situation (software option 136)	720

▷

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

3

Using Fixed Cycles

3.1 Working with fixed cycles

Machine-specific cycles

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

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

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

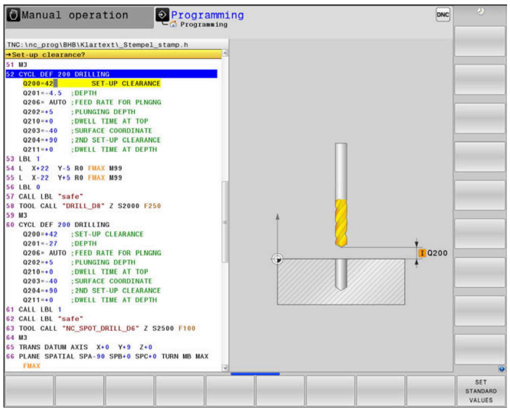
Some machine-specific cycles use transfer parameters that are also part of HEIDENHAIN standard cycles. In order to avoid problems (related to overwriting of transfer parameters that are used more than once), when using DEF-active cycles (cycles that the control runs automatically during cycle definition)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

Further information: "Calling a cycle", Page 60

do the following: The following procedure is advisable:

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

Defining a cycle using soft keys



CYCL DEF

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

DRILLING/ THREAD

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

262

- Select the cycle, e.g. **THREAD MILLING**. The control initiates a dialog and prompts you for all required input values. At the same time, a graphic is displayed in the right half of the screen. The required parameter is highlighted
- Enter all parameters required by the control. Conclude each input with the **ENT** key
- The control closes the dialog when all required data has been entered

Defining a cycle using the GOTO function

CYCL DEF

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

GOTO □

- The control opens the smartSelect selection window with an overview of the cycles
- Select the desired cycle with the cursor keys or the mouse. The control then initiates the cycle dialog as described above

Example

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

Calling a cycle

Requirements

Before calling a cycle, be sure to program:

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

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

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

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

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

Calling a cycle with CYCL CALL

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

CYCL CALL

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

Calling a cycle with CYCL CALL PAT

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

Further information: "Pattern definition with PATTERN DEF", Page 68

Further information: "Point tables", Page 75

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**, program the following:

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

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

Calling a cycle with SEL CYCLE

By pressing the **PGM CALL** key and entering SELECT CYCLE, you can call any desired NC program as a machining cycle. The system displays the syntax for **SEL CYCLE** , and you can select the desired NC program with SELECT FILE. This program can then be called with **CYCLE CALL**, **CYCL CALL PAT**, **CYCL CALL POS**, or 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.

Working with a parallel axis

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

This is only possible when programming the following cycles:

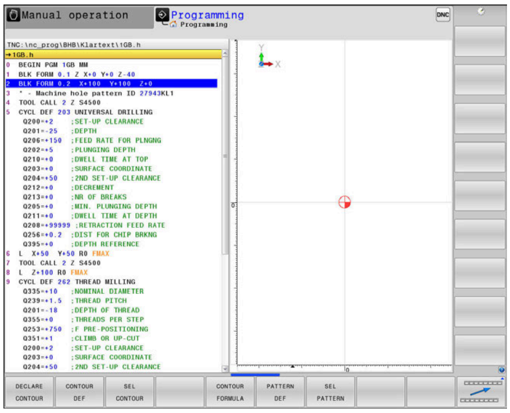
Cycle	W axis function
200 **DRILLING**	■
201 **REAMING**	■
202 **BORING**	■
203 **UNIVERSAL DRILLING**	■
204 **BACK BORING**	■
205 **UNIVERSAL PECKING**	■
208 **BORE MILLING**	■
225 **ENGRAVING**	■
232 **FACE MILLING**	■
233 **FACE MILLING**	■
241 **SINGLE-LIP D.H.DRLNG**	■

HEIDENHAIN recommends not to use **TOOL CALL W**! Use **FUNCTION PARAXMODE** or **FUNCTION PARAXCOMP**.

Further information: User's Manual for Conversational Programming

3.2 Program defaults for cycles

Overview

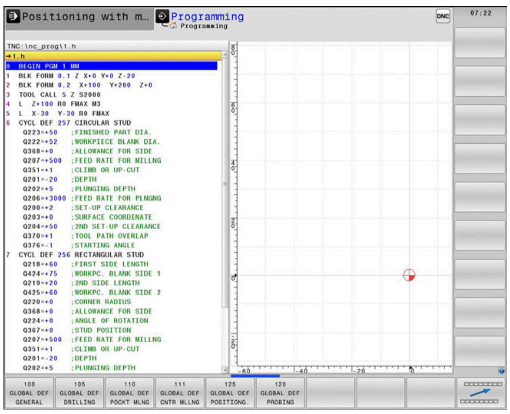


All Cycles 20 to 25, as well as all of those with numbers 200 or higher, always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** you can define these cycle parameters at the beginning of the program, so that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

The following GLOBAL DEF functions are available:

Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	66
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	66
110 GLOBAL DEF POCKT MLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	66
111 GLOBAL DEF CNTR MLLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	66
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	67
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	67

Entering GLOBAL DEF



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

SPEC FCT

- Press the **SPEC FCT** key to select the special functions

PROGRAM DEFAULTS

- Select the functions for program defaults

GLOBAL DEF

- Press the **GLOBAL DEF** soft key

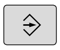
100 GLOBAL DEF GENERAL

- Select the desired GLOBAL DEF function, e.g. by pressing the **GLOBAL DEF GENERAL** soft key
- Enter the required definitions, and confirm each entry with the **ENT** key

Using GLOBAL DEF information

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

Proceed as follows:



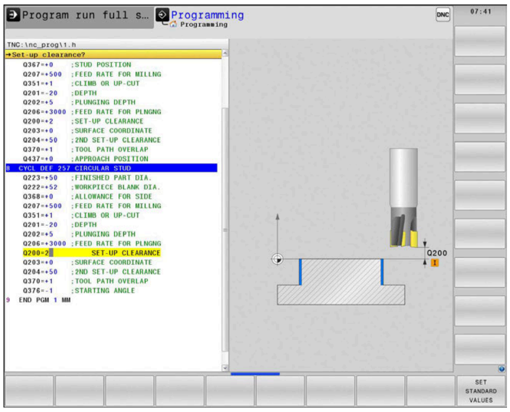
- Operating mode: Press the **Programming** key

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

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

- Select the desired cycle, e.g. **drilling**
- If a global parameter exists, the control will display the **SET STANDARD VALUES** soft key

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



NOTICE

Danger of collision!

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

- Make sure to use **GLOBAL DEF** carefully. Test your program before executing it
- If a fixed value is entered in machining cycles, **GLOBAL DEF** does not modify this value

Global data valid everywhere

- **Safety clearance**: Distance between tool face and workpiece surface for automated approach of the cycle start position in the tool axis
- **2nd set-up clearance:** Position to which the control positions the tool at the end of a machining step. The next machining position is approached at this height in the working plane
- **F positioning:** Feed rate at which the control traverses the tool within a cycle
- **F retraction:** Feed rate at which the control retracts the tool

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

Global data for drilling operations

- **Retraction rate for chip breaking:** Value by which the control retracts the tool during chip breaking
- **Dwell time at depth**: Time in seconds that the tool remains at the hole bottom
- **Dwell time at top**: Time in seconds that the tool remains at the set-up clearance.

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

Global data for milling operations with pocket cycles 25x

- **Overlap factor**: The tool radius multiplied by the overlap factor equals the stepover
- **Climb or up-cut**: Select the type of milling
- **Plunging type**: Plunge into the material helically, in a reciprocating motion, or vertically

> The parameters apply to milling cycles 251 to 257.

Global data for milling operations with contour cycles

- **Set-up clearance**: Distance between tool face and workpiece surface for automated approach of the cycle start position in the Tool axis
- **Clearance height**: Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle)
- **Overlap factor**: The tool radius multiplied by the overlap factor equals the stepover
- **Climb or up-cut**: Select the type of milling

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

Global data for positioning behavior

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

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

Global data for probing functions

- **Set-up clearance**: Distance between stylus and workpiece surface for automated approach of the probing position
- **Clearance height:** The coordinate in the touch probe axis to which the control traverses the touch probe between measuring points if the **Move to clearance height** option is activated
- **Move to clearance height**: Select whether the control moves the touch probe to the set-up clearance or clearance height between the measuring points

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

3.3 Pattern definition with PATTERN DEF

Application

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

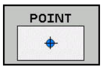
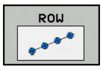
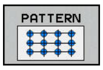
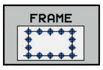
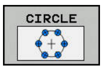
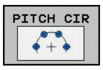
NOTICE

Danger of collision!

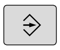
The **PATTERN DEF** function calculates the machining coordinates in the **X** and **Y** axes For all tools axes apart from **Z** there is a danger of collision in the following operation!

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

The following machining patterns are available:

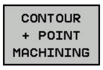
Soft key	Machining pattern	Page
	POINT Definition of up to any 9 machining positions	70
	ROW Definition of a single row, straight or rotated	70
	PATTERN Definition of a single pattern, straight, rotated or distorted	71
	FRAME Definition of a single frame, straight, rotated or distorted	72
	CIRCLE Definition of a full circle	73
	PITCH CIRCLE Definition of a pitch circle	74

Entering PATTERN DEF

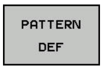


- Operating mode: Press the **Programming** key

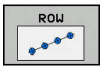
- Press the **SPEC FCT** key to select the special functions



- Select the functions for contour and point machining



- Press the **PATTERN DEF** soft key



- Select the desired machining pattern, e.g. press the "single row" soft key
- Enter the required definitions, and confirm each entry with the **ENT** key

Using PATTERN DEF

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

Further information: "Calling a cycle", Page 60

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

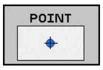
> A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.
>
> You can use the mid-program startup function to select any point at which you want to start or continue machining.
> **Further information**: User's Manual for Setup, Testing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

Defining individual machining positions

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

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

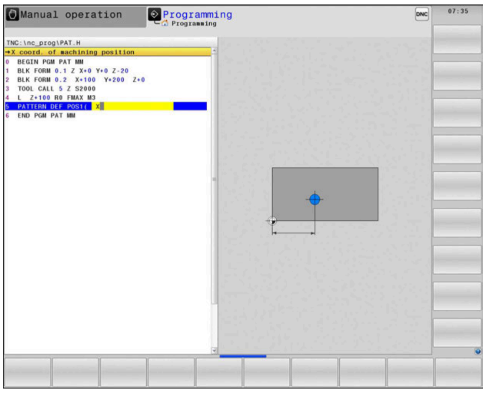
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- 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

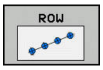
Example

```
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

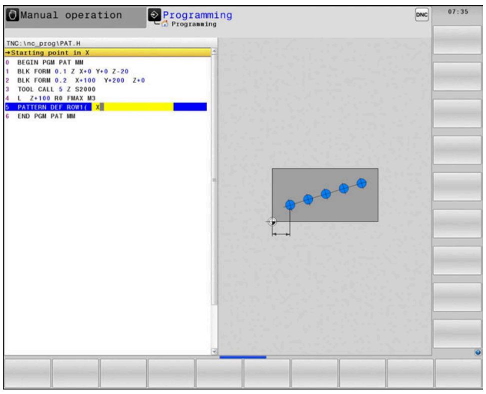
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **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

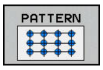
Example

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



Defining a single pattern

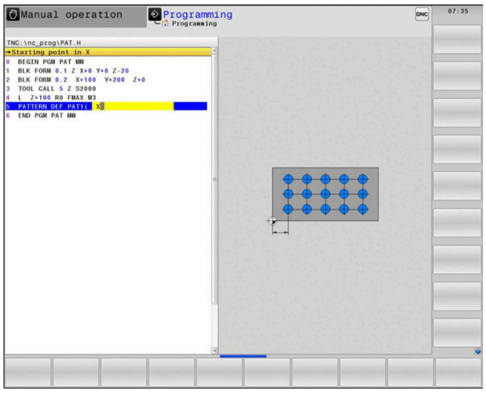
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.
>
> The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position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



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

Example

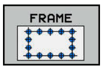
```
10 L  Z+100 R0  FMAX
11 PATTERN DEF  PAT1 (X+25 Y+33,5
   DX+8 DY+10  NUMX5 NUMY4 ROT+0
   ROTX+0 ROTY+0  Z+0)
```



Defining individual frames

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

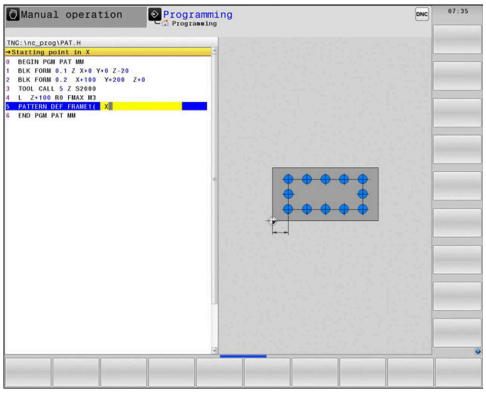
The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.



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

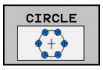
Example

```
10 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

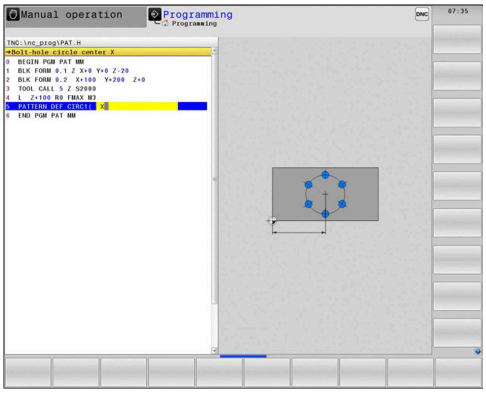
If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **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

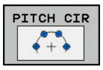
Example

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



Defining a pitch circle

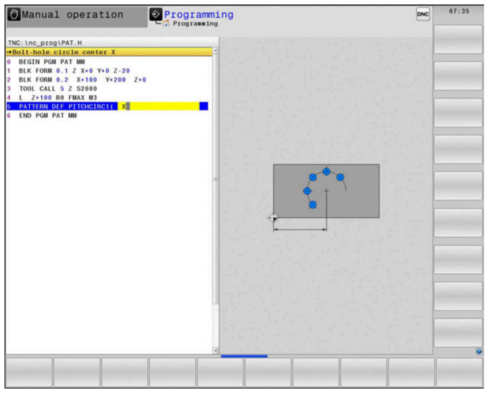
> If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- **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

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

PGM MGT

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

FILE NAME?

ENT

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

MM

- Select the unit of measure: Press the **MM** or **INCH** soft key. The control changes to the program window and displays an empty points table

INSERT LINE

- Press the **INSERT LINE** soft key to insert a new line. Enter the coordinates of the desired machining position

Repeat the process until all desired coordinates have been entered.

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

Hiding single points from the machining process

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

- In the table, select the point to be hidden
- Select the **FADE** column
- Activate hiding or
- Deactivate hiding

Selecting a point table in the NC program

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

PGM CALL

- Press the **PGM CALL** key to call the function for selecting the point table

SELECT POINT TABLE

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

SELECT FILE

- Press the **SELECT FILE** soft key
- Select the point table and confirm with the **OK** soft key

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

Example

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

Calling a cycle in connection with point tables

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

CYCL CALL

- To program the cycle call: Press the **CYCL CALL** key
- To call the point table, press the **CYCL CALL PAT** soft key
- Enter the feed rate at which the control is to move from point to point or press the **F MAX** soft key (if you make no entry, the control will move at the last programmed feed rate)
- Enter a miscellaneous function (M function) if required. Confirm your input with the **END** key

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

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

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

Effect of the point table with SL cycles and Cycle 12

The control interprets the points as an additional datum shift.

Effect of the point table with Cycles 200 to 208, and 262 to 267

The control interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table as the starting point coordinate in the spindle axis, you must define the coordinate of the workpiece upper edge (Q203) as 0.

Effect of the point table with Cycles 251 to 254

The control interprets the points on the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table as the starting point coordinate in the spindle axis, you must define the coordinate of the workpiece upper edge (Q203) as 0.

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

NOTICE

Danger of collision!

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

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

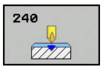
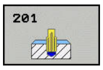
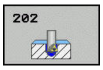
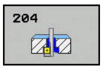
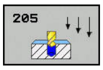
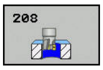
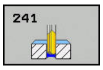
4

Fixed Cycles: Drilling

4.1 Fundamentals

Overview

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

Soft key	Cycle	Page
	240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth	81
	200 DRILLING With automatic pre-positioning, 2nd set-up clearance	83
	201 REAMING With automatic pre-positioning, 2nd set-up clearance	85
	202 BORING With automatic pre-positioning, 2nd set-up clearance	87
	203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	90
	204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	96
	205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	100
	208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance	108
	241 SINGLE-LIP D.H.DRLNG With automatic pre-positioning to deepened starting point, shaft speed and coolant definition	111

4.2 CENTERING (Cycle 240, ISO: G240)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface
2 The tool is centered at the programmed feed rate **F** to the programmed centering diameter or centering depth.
3 If defined, the tool remains at the centering depth.
4 Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

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

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

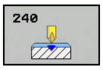
NOTICE

Danger of collision!

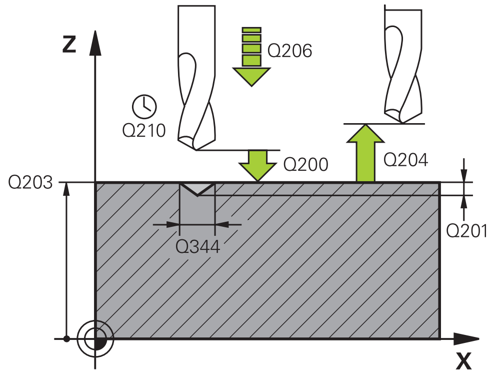
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

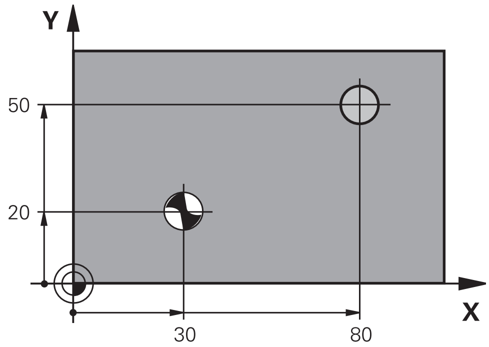
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range: 0 to 99999.9999
- **Q343 Select diameter/depth (1/0)**: Select whether centering is based on the entered diameter or depth. If the control is to center based on the entered diameter, the point angle of the tool must be defined in the **T-angle** column of the TOOL.T tool table.
  0: Centering based on the entered depth
  1: Centering based on the entered diameter
- **Q201 Depth?** (incremental): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined. Input range: -99999.9999 to 99999.9999
- **Q344 Diameter of counterbore** (algebraic sign): Centering diameter. Only effective if Q343=1 is defined. Input range: -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during centering. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





Example

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

4.3 DRILLING (Cycle 200)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 The tool drills to the first plunging depth at the programmed feed rate **F**
3 The Control retracts the tool at **FMAX** to the set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to the set-up clearance above the first plunging depth.
4 The tool then drills deeper by the plunging depth at the programmed feed rate F.
5 The control repeats this process (2 to 4) until the programmed depth is reached (the dwell time from Q211 is effective with every infeed)
6 Finally, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

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

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

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

NOTICE

Danger of collision!

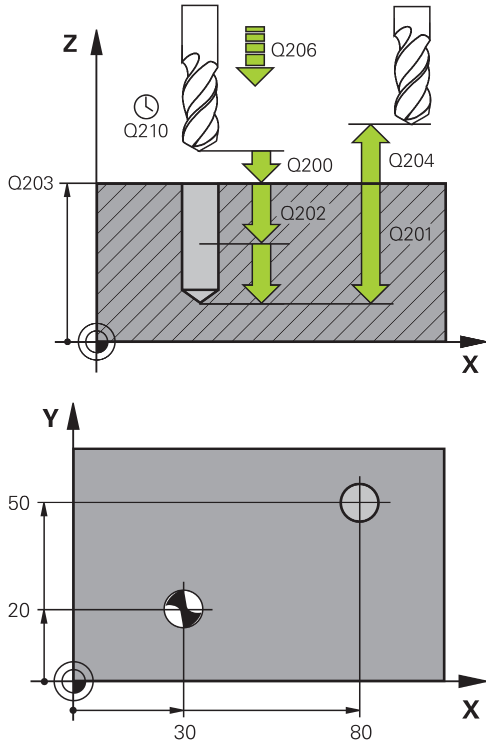
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters

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

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



Example

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

4.4 REAMING (Cycle 201, ISO: G201)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 The tool reams to the entered depth at the programmed feed rate **F**.
3 If programmed, the tool remains at the hole bottom for the entered dwell time.
4 Then, the control retracts the tool at rapid traverse **FMAX** to setup clearance or to the 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

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

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

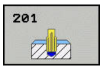
NOTICE

Danger of collision!

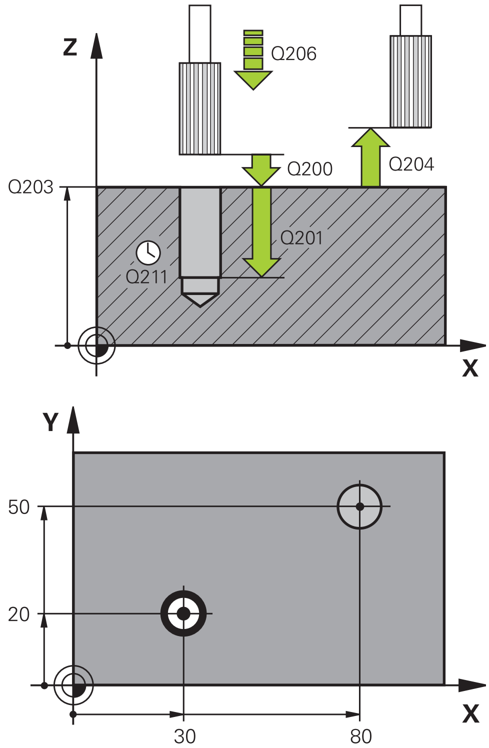
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during reaming. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for reaming applies. Input range: 0 to 99999.999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range: 0 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



Example

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

4.5 BORING (Cycle 202, ISO: G202)

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface
2. The tool drills to the programmed depth at the feed rate for plunging.
3. If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4. The control then carries out an oriented spindle stop to the position that is defined in the **Q336** parameter
5. If retraction is selected, the control retracts in the programmed direction by 0.2 mm (fixed value)
6. The tool then retracts to set-up clearance at the retraction rate, and from there —if programmed—to the 2nd set-up clearance at **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**. If **Q214**=0 the tool tip remains on the wall of the hole
7. The control then returns the tool to the center of the hole

Please note while programming:

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

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

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

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

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

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

NOTICE

Danger of collision!

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

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

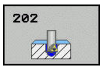
NOTICE

Danger of collision!

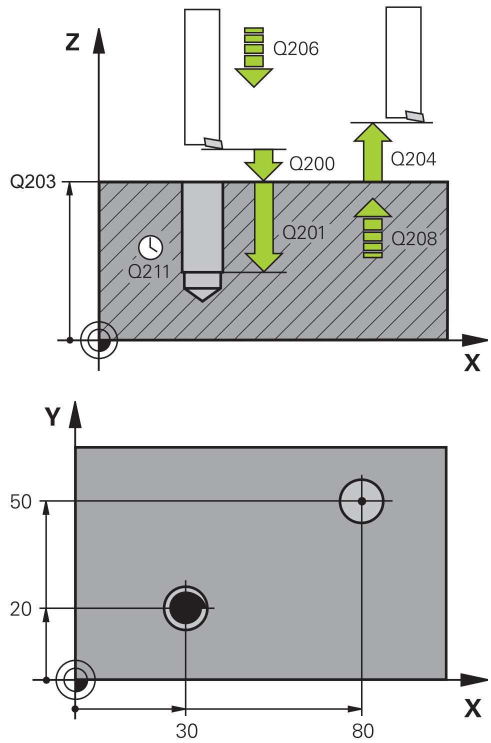
There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

- Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during boring. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the feed rate for plunging applies. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q214 Disengaging directn (0/1/2/3/4)?**: Determine the direction in which the control retracts the tool at the hole bottom (after carrying out an oriented spindle stop)
  0: Do not disengage the tool
  1: Disengage the tool in the minus direction of the reference axis
  2: Disengage the tool in the minus direction of the minor axis
  3: Disengage the tool in the plus direction of the reference axis
  4: Disengage the tool in the plus direction of the minor axis
- **Q336 Angle for spindle orientation?** (absolute): Angle to which the control positions the tool before retracting it. Input range: -360.000 to 360.000



Example

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

4.6 UNIVERSAL DRILLING (Cycle 203, ISO: G203)

Cycle run

Behavior without chip breaking and without decrement:

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
2. The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3. Then, the control retracts the tool from the hole to **SET-UP CLEARANCE Q200**
4. Now, the control again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTHQ202 FEED RATE FOR PLNGNG Q206**
5. When machining without chip breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and remains there for the **DWELL TIME AT TOP Q210**.
6. This procedure is repeated until **depth Q201** is achieved.
7. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and without decrement:

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
2. The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3. Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4. Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** at **FEED RATE FOR PLNGNG Q206**
5. The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
6. If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**
7. Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8. Steps 2 to 7 will be repeated until **DEPTH Q201** is reached.
9. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking and with decrement

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified **SAFETY CLEARANCE Q200** above the workpiece surface
2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** minus **DECREMENT Q212** at **FEED RATE FOR PLNGNG Q206**. The increasingly smaller difference between the updated **PLUNGING DEPTH Q202** minus **DECREMENT Q212** must never be smaller than **MIN. PLUNGING DEPTH Q205** (example: **Q202**=5, **Q212**=1, **Q213**=4, **Q205**= 3: The first plunging depth is 5 mm, the second plunging depth is 5 - 1 = 4 mm, the third plunging depth is 4 - 1 = 3 mm, the fourth plunging depth is also 3 mm)
5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
6 If programmed, the control will now wait for the time specified in **DWELL TIME AT TOP Q210**
7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached.
9 If programmed, the control will now wait for the time specified in **DWELL TIME AT DEPTH Q211**
10 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE** The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Please note while programming:

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

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

NOTICE

Danger of collision!

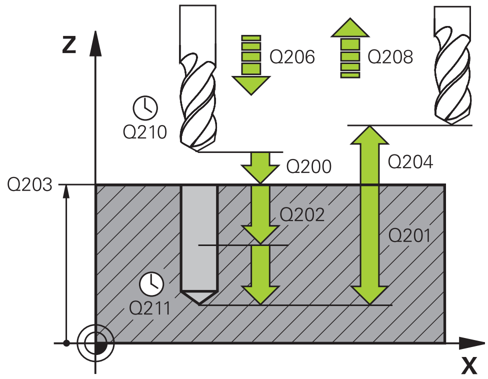
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters

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

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q210 Dwell time at the top?**: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal by the Control. Input range: 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999
- **Q213 Nr of breaks before retracting?**: Number of chip breaks before the control will retract the tool from the hole for chip removal. For chip breaking, the control retracts the tool each time by the value in **Q256**. Input range: 0 to 99999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999



Example

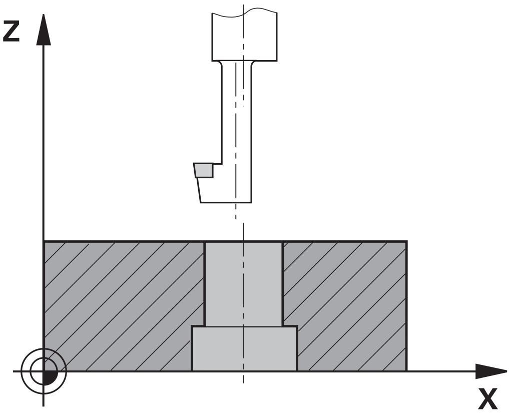
11 CYCL DEF 203 UNIVERSAL DRILLING	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q211=0	;DWELL TIME AT TOP
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q212=0.2	;DECREMENT
Q213=3	;NR OF BREAKS
Q205=3	;MIN. PLUNGING DEPTH
Q211=0.25	;DWELL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

- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the control retracts the tool at the feed rate specified in **Q206**. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the TOOL.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool

4.7 BACK BORING (Cycle 204, ISO: G204)

Cycle run

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



1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface
2. The control then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.
3. The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the cutting edge has reached programmed set-up clearance beneath the lower workpiece edge
4. The control then centers the tool again in the bore hole, switches on the spindle and the coolant and moves at the feed rate for counterboring to the depth programmed for the counterbore
5. If programmed, the tool remains at the counterbore bottom. The tool will then be retracted from the hole again. The control carries out another oriented spindle stop and the tool is once again displaced by the off-center distance
6. Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**
7. The control then returns the tool to the center of the hole

Please note while programming:

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

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

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

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

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

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

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

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

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

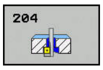
NOTICE

Danger of collision!

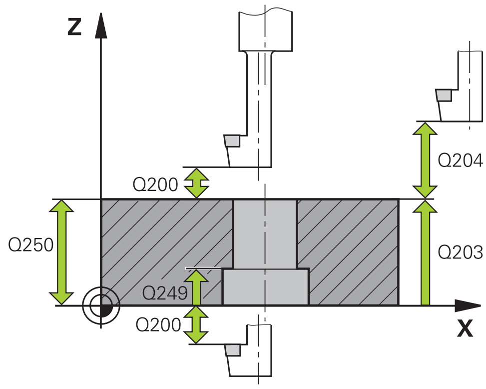
There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

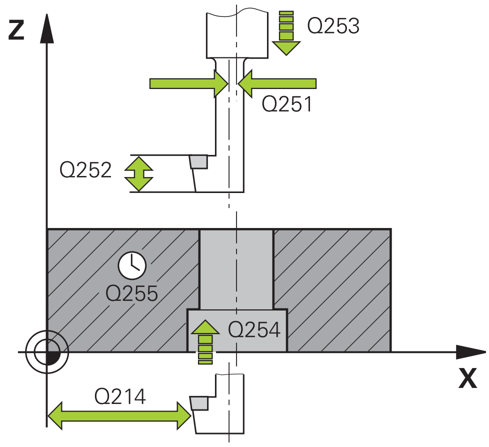
- Check the position of the tool tip when you program an oriented spindle stop with reference to the angle that you enter in **Q336** (e.g. in the **Positioning w/ Manual Data Input** mode of operation). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Select the disengaging direction Q214 so that the tool moves away from the edge of the hole

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q249 Depth of counterbore?** (incremental): Distance between underside of workpiece and bottom of the hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range: -99999.9999 to 99999.9999
- **Q250 Material thickness?** (incremental): Thickness of the workpiece. Input range: 0.0001 to 99999.9999
- **Q251 Tool edge off-center distance?** (incremental): Off-center distance for the boring bar; value from the tool data sheet. Input range: 0.0001 to 99999.9999
- **Q252 Tool edge height?** (incremental): Distance between the underside of the boring bar and the main cutting edge; value from tool data sheet. Input range: 0.0001 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Q255 Dwell time in secs.?**: Dwell time in seconds at the bottom of the bore hole. Input range: 0 to 3600.000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





Example

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

- **Q214 Disengaging directn (0/1/2/3/4)?**: Determine the direction in which the control will displace the tool by the off-center distance (after having carried out an oriented spindle stop); programming 0 is not allowed
  1: Retract the tool in the negative direction of the reference axis
  2: Retract the tool in the negative direction of the minor axis
  3: Retract the tool in the positive direction of the reference axis
  4: Retract the tool in the positive direction of the minor axis
- **Q336 Angle for spindle orientation?** (absolute): Angle at which the control positions the tool before it is plunged into or retracted from the bore hole. Input range: -360.0000 to 360.0000

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

4.8 UNIVERSAL PECKING (Cycle 205, ISO: G205)

Cycle run

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2. If you enter a deepened starting point, the control moves at the defined positioning feed rate to the set-up clearance above the recessed starting point
3. The tool drills to the first plunging depth at the programmed feed rate **F**
4. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered advanced stop distance above the first plunging depth
5. The tool then drills deeper by the plunging depth at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
6. The control repeats this procedure (steps 2 to 4) until the total hole depth is reached
7. The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to set-up clearance or the 2nd set-up clearance at the retraction feed rate. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

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

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

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

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

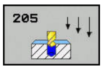
NOTICE

Danger of collision!

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

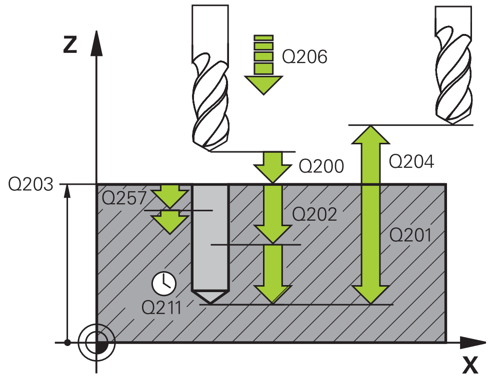
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters



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

  The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the control decreases the **Q202** plunging depth. Input range: 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999
- **Q258 Upper advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole. Input range 0 to 99999.9999
- **Q259 Lower advanced stop distance?** (incremental): Set-up clearance for rapid traverse positioning when the control returns the tool to the current plunging depth after having retracted it from the hole; value for the last plunging depth. Input range: 0 to 99999.9999



Example

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

- **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range: 0 to 3600.0000
- **Q379 Deepened starting point?** (incremental, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed starting point. Input range: 0 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q206**. Input range: 0 to 99999,9999; alternatively **FMAX, FAUTO**
- **Q395 Diameter as reference (0/1)?**: Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T ANGLE** column of the TOOL.T tool table.
  0 = Depth references the tool tip
  1 = Depth references the cylindrical part of the tool

Position behavior when working with Q379

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

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

Start of drilling

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

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

Example:

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

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

Start of drilling at deepened starting point

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

Chip breaking

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

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

Example:

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

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

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

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

4.9 BORE MILLING (Cycle 208)

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface. Then, it moves the tool on a circular arc with the indicated diameter (provided that enough space is available)
2 The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate **F**.
3 When the drilling depth is reached, the control once again traverses a full circle to remove the material remaining after the initial plunge.
4 The control then positions the tool at the center of the hole again.
5 Finally, the tool is retracted to the setup clearance or to the 2nd setup clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Please note while programming:

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

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

If you have entered the bore hole diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation.

An active mirror function **does not** influence the type of milling defined in the cycle.

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

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

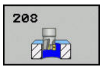
NOTICE

Danger of collision!

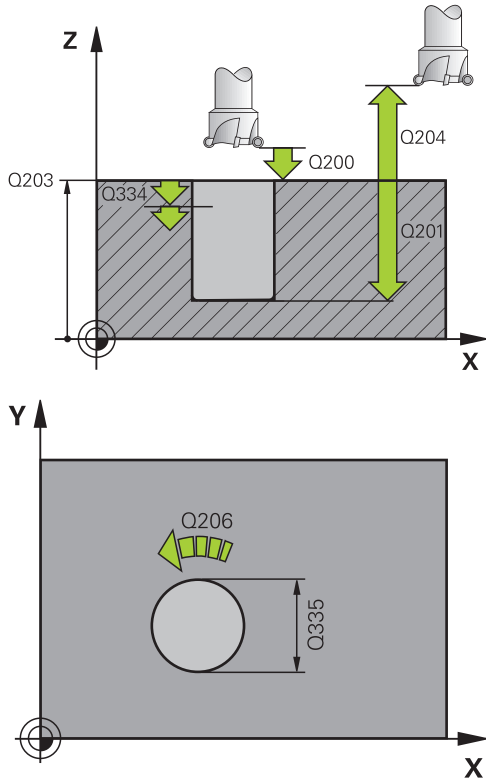
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between underside of tool and the workpiece surface. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during helical drilling. Input range: 0 to 99999.999; alternatively **FAUTO**, **FU**, **FZ**
- **Q334 Feed per revolution of helix** (incremental): Depth of the tool plunge with each helix (=360°). Input range: 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q335 Nominal diameter?** (absolute): Hole diameter. If you entered the nominal diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation. Input range: 0 to 99999.9999
- **Q342 Roughing diameter?** (absolute): As soon as you enter a value greater than 0 in Q342, the control no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter. Input range: 0 to 99999.9999
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (if you enter 0, climb milling is performed)



Example

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

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

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 104, 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 104
3. The control executes the approach motion depending on the direction of rotation defined in the cycle with a spindle that rotates clockwise, counterclockwise, or is stationary
4. The tool drills to the hole depth at the feed rate **F**, or to the maximum plunging depth if a smaller infeed value has been entered. The plunging depth is decreased after each infeed by the decrement. If you have entered a dwell depth, the control reduces the feed rate by the feed rate factor after the dwell depth has been reached
5. If programmed, the tool remains at the hole bottom for chip breaking.
6. The control repeats this procedure (steps 4 to 5) until the total hole depth is reached
7. After the control has reached this position, it will automatically switch off the coolant as soon as the speed has reached the value defined in Q427 **ROT.SPEED INFEED/OUT**
8. The control positions the tool to the retract position at the retraction feed rate. To find out the retract position value in your particular case, please refer to: see Page 104
9. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**

Please note while programming:

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

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

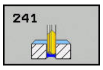
NOTICE

Danger of collision!

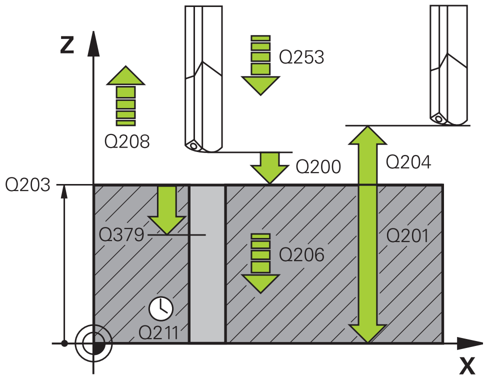
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and **Q203 SURFACE COORDINATE**. Input range: 0 to 99999.9999
- **Q201 Depth?** (incremental): Distance between **Q203 SURFACE COORDINATE** and bottom of hole. Input range: -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during drilling. Input range 0 to 99999.999, alternatively **FAUTO**, **FU**
- **Q211 Dwell time at the depth?**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Distance to workpiece datum. Input range: -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q379 Deepened starting point?** (incremental, references **Q203 SURFACE COORDINATE**, takes **Q200** into account): Starting position of actual drilling. The control moves at **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** above the recessed starting point. Input range: 0 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208**=0, the control retracts the tool at **Q206 FEED RATE FOR PLNGNG**. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- **Q426 Rot. dir. of entry/exit (3/4/5)?**: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input:
  3: Turn the spindle with M3
  4: Turn the spindle with M4
  5: Move with stationary spindle
- **Q427 Spindle speed of entry/exit?**: Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input range: 0 to 99999
- **Q428 Spindle speed for drilling?**: Desired speed for drilling. Input range: 0 to 99999



Example

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

- **Q429 M function for coolant on?**: Miscellaneous function for switching on the coolant. The control switches on the coolant on if the tool is in the hole at **Q379 STARTING POINT**. Input range: 0 to 999
- **Q430 M function for coolant off?**: Miscellaneous function for switching off the coolant. The control switches the coolant off if the tool is at **Q201 DEPTH**. Input range: 0 to 999
- **Q435 Dwell depth?** (incremental): Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (default setting). Application: During machining of through-holes some tools require a short dwell time before leaving the bottom of the hole in order to transport the chips to the top. Define a value smaller than **Q201 DEPTH**; input range: 0 to 99999.9999
- **Q401 Feed rate factor in %?**: Factor by which the control reduces the feed rate after **Q435 DWELL DEPTH** has been reached. Input range: 0 to 100
- **Q202 Maximum plunging depth?** (incremental): Infeed per cut. **Q201 DEPTH** does not have to be a multiple of **Q202**. Input range 0 to 99999.9999
- **Q212 Decrement?** (incremental): Value by which the control decreases **Q202 Feed depth** after each infeed. Input range: 0 to 99999.9999
- **Q205 Minimum plunging depth?** (incremental): If you have entered **Q212 DECREMENT**, the control limits the plunging depth to the value for **Q205**. Input range: 0 to 99999.9999

Position behavior when working with Q379

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

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

Start of drilling

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

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

Example:

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

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

Start of drilling at deepened starting point

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

Chip breaking

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

STARTING POINT Q379=0

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

STARTING POINT Q379>0

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

Example:

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

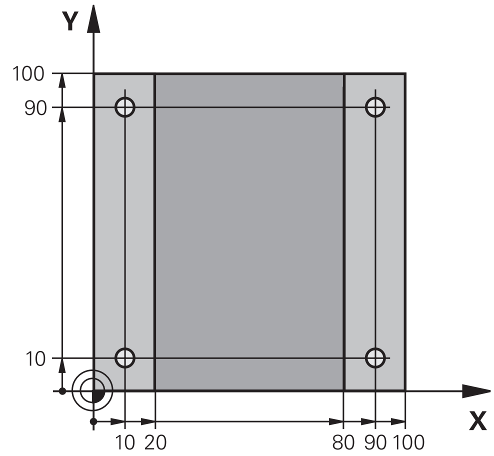
The following table shows various examples for calculating the position for chip breaking (retraction position):

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

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

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	

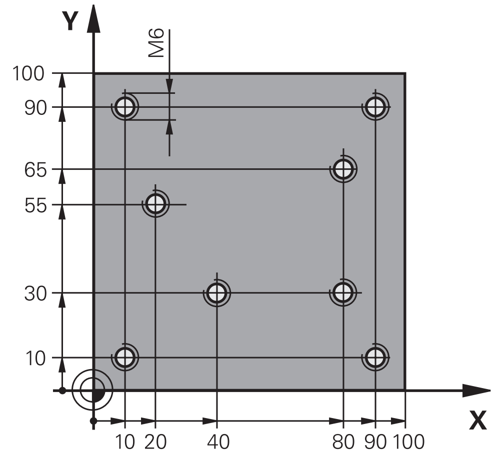
Example: Using drilling cycles in connection with PATTERN DEF

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 124



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	

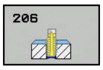
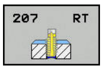
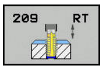
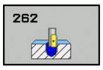
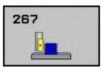
5

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	125
	207 TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	128
	209 TAPPING WITH CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	133
	262 THREAD MILLING Cycle for milling a thread in pre-drilled material	140
	263 THREAD MILLING/ COUNTERSINKING Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	144
	264 THREAD DRILLING/ MILLING Cycle for drilling into solid material with subsequent milling of the thread with a tool	148
	265 HELICAL THREAD DRILLING/MILLING Cycle for milling the thread into solid material	152
	267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer	156

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:

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

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

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

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

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

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

The spindle speed potentiometer is inactive.

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

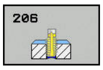
NOTICE

Danger of collision!

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

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

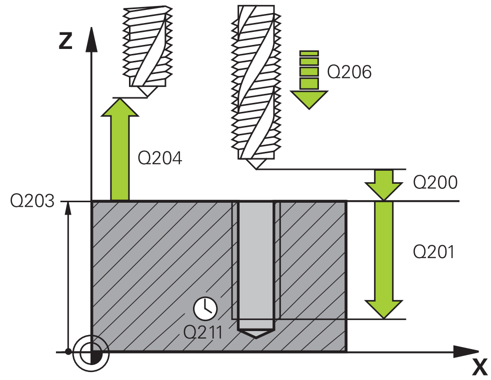
Cycle parameters



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

Guide value: 4x pitch.

- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during tapping. Input range 0 to 99999.999 alternatively **FAUTO**
- **Q211 Dwell time at the depth?**: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



Example

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

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

F: Feed rate (mm/min)

S: Spindle speed (rpm)

p: Thread pitch (mm)

Retracting after a program interruption

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

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

Cycle run

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

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2. The tool drills to the total hole depth in one movement.
3. It then reverses the direction of spindle rotation and the tool is retracted to the set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**
4. The control stops the spindle turning at that set-up clearance

Please note while programming:

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

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

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

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

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

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

The spindle speed potentiometer is inactive.

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

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

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

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

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

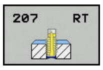
NOTICE

Danger of collision!

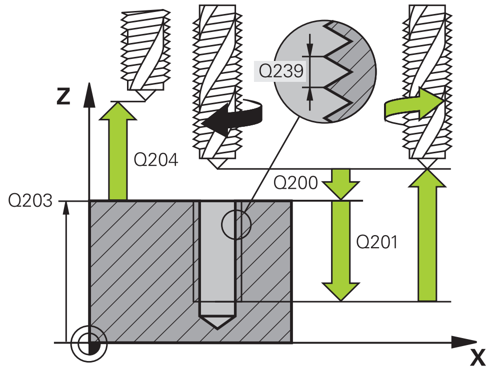
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters



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



Example

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

Retracting after a program interruption

Retracting in the Manual Operation mode

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

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

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

NOTICE

Danger of collision!

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

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

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

Cycle run

The tool machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking or not.

1. The control positions the tool in the tool axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface.There, it carries out an oriented spindle stop
2. The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition. If you have defined a factor for increasing the spindle speed, the control retracts from the hole at the corresponding speed
3. It then reverses the direction of spindle rotation again and advances to the next infeed depth.
4. The control repeats this procedure (steps 2 to 3) until the programmed thread depth is reached
5. The tool is then retracted to set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**
6. The control stops the spindle turning at that set-up clearance

Please note while programming:

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

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

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

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

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

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

The spindle speed potentiometer is inactive.

If you defined a speed factor for fast retraction in cycle parameter **Q403**, the control limits the speed to the maximum speed of the active gear stage.

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

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

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

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

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

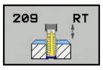
NOTICE

Danger of collision!

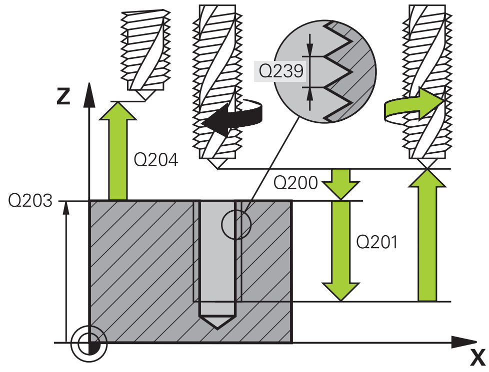
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters



- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  –= left-hand thread
  Input range: -99.9999 to +99.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Q256 Retract dist. for chip breaking?**: The control multiplies the pitch **Q239** by the programmed value and retracts the tool by the calculated value for chip breaking. If you enter **Q256** = 0, the control retracts the tool completely from the hole (to the set-up clearance) for chip breaking. Input range 0.000 to 99999.999
- **Q336 Angle for spindle orientation?** (absolute): Angle to which the control positions the tool before machining the thread. This allows you to recut the thread, if required. Input range -360.0000 to 360.0000
- **Q403 RPM factor for retraction?**: Factor by which the control increases the spindle speed—and therefore also the retraction feed rate—when retracting from the hole. Input range 0.0001 to 10. Maximum increase to maximum speed of the active gear stage.



Example

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

Retracting after a program interruption

Retracting in the Manual Operation mode

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

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

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

NOTICE

Danger of collision!

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

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

5.5 Fundamentals of Thread Milling

Prerequisites

- Your machine tool features internal spindle cooling (cooling lubricant at least 30 bars, compressed air supply at least 6 bars)
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius **DR** in the **TOOL CALL**
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265 you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread / – = left-hand thread) and milling method Q351 (+1 = climb / –1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

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

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

NOTICE

Danger of collision!

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

- Make sure to program all depth values with the same algebraic sign. Example: If you program the Q356 COUNTERSINKING DEPTH parameter with a negative sign, then Q201 DEPTH OF THREAD must also have a negative sign
- If you would like to repeat just the counterbore procedure in a cycle, you can enter 0 for DEPTH OF THREAD. In this case, the machining direction is determined by the programmed COUNTERSINKING DEPTH

NOTICE

Danger of collision!

A collision may occur if, upon tool breakage, you retract the tool from the hole in the direction of the tool axis only.

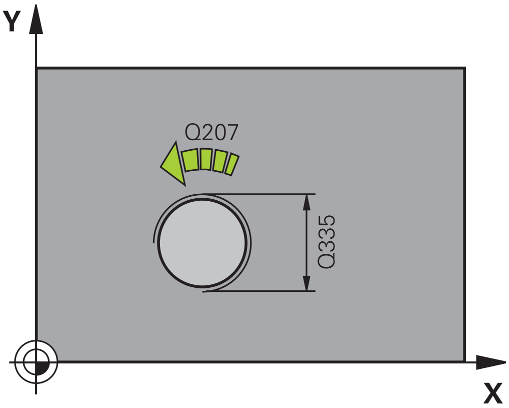
- Stop the program run if the tool breaks
- Change to Positioning with Manual Data Input mode of operation
- First move the tool in a linear movement towards the hole center
- Retract the tool in the tool axis direction

The programmed feed rate for thread milling references the cutting edge of the tool. However, since the control always displays the feed rate relative to the center path of the tool tip, the displayed value does not match the programmed value.

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

5.6 THREAD MILLING (Cycle 262, ISO: G262)

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:

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

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

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

The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the pitch of the tool diameter is four times smaller than the nominal thread diameter.

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

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

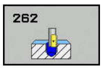
NOTICE

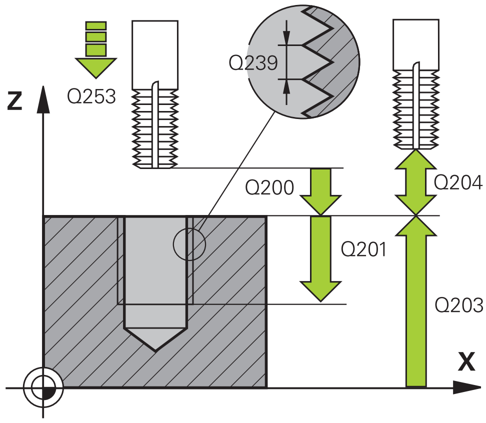
Danger of collision!

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

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

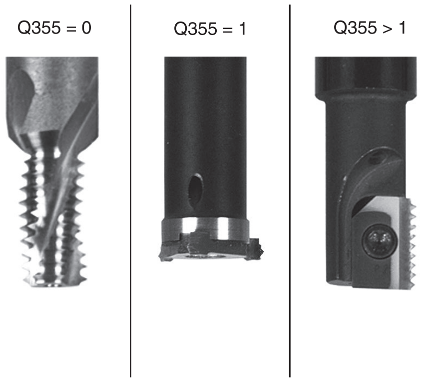
Cycle parameters





- **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  –= left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- **Q355 Number of threads per step?**: Number of turns by which the tool is displaced:
  0 = one helix over the entire thread depth
  1 = continuous helix over the entire thread length
  >1 = several helix paths with approach and departure, between these, the control offsets the tool by **Q355** x pitch. Input range 0 to 99999

- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (if you enter 0, climb milling is performed)
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range 0 to 99999.999 alternatively **FAUTO**



Example

25 CYCL DEF 262 THREAD MILLING	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-20	;DEPTH OF THREAD
Q355=0	;THREADS PER STEP
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q207=500	;FEED RATE FOR MILLNG
Q512=0	;FEED FOR APPROACH

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

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:

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

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

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

If you want to countersink at front, define the countersinking depth as 0.

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

NOTICE

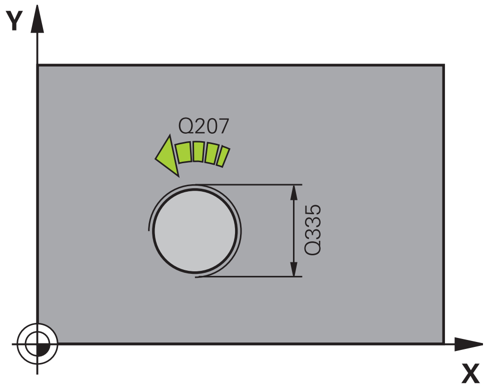
Danger of collision!

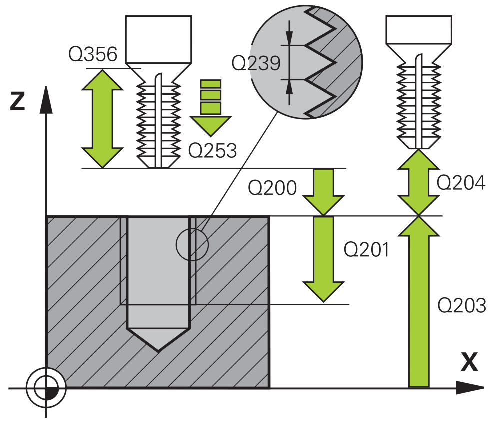
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

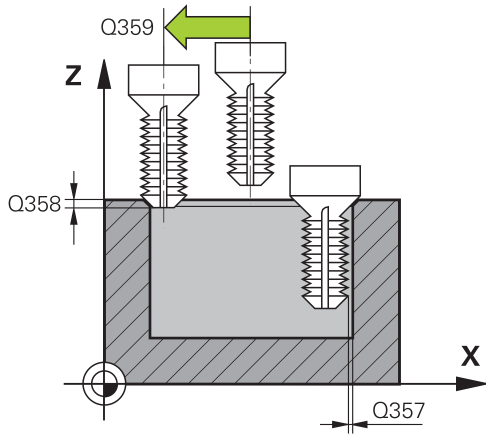
Cycle parameters

- **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  –= left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- **Q356 Countersinking depth?** (incremental): Distance between workpiece surface and tool tip. Input range -99999.9999 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (if you enter 0, climb milling is performed)
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q357 Safety clearance to the side?** (incremental): Distance between the cutting edge and the wall of the hole. Input range 0 to 99999.9999
- **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range 0 to 99999.9999





- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range 0 to 99999.999 alternatively **FAUTO**



Example

25 CYCL DEF 263 THREAD MLLNG/ CNTSNKG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-16	;DEPTH OF THREAD
Q356=-20	;COUNTERSINKING DEPTH
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q200=2	;SET-UP CLEARANCE
Q357=0.2	;CLEARANCE TO SIDE
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLNG
Q512=0	;FEED FOR APPROACH

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

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:

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

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

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

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

NOTICE

Danger of collision!

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

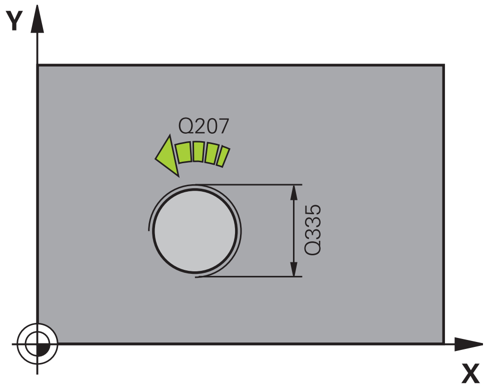
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

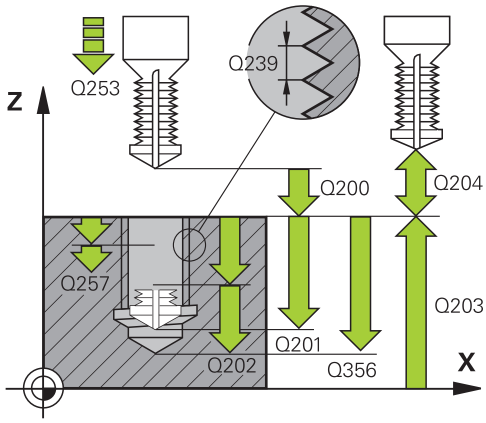
Cycle parameters

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

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

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





Example

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

- **Q257 Infeed depth for chip breaking?** (incremental): Plunging depth after which the control breaks the chip. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- **Q256 Retract dist. for chip breaking?** (incremental): Value by which the control retracts the tool during chip breaking. Input range 0.000 to 99999.999
- **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range 0 to 99999.999 alternatively **FAUTO**

Q202=5	;PLUNGING DEPTH
Q258=0.2	;UPPER ADV STOP DIST
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=0.2	;DIST FOR CHIP BRKNG
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q206=150	;FEED RATE FOR PLNGNG
Q207=500	;FEED RATE FOR MILLNG
Q512=0	;FEED FOR APPROACH

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

Cycle run

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

Countersinking at front

2 If countersinking occurs before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking occurs after thread milling, the control moves the tool to the countersinking depth at the feed rate for prepositioning
3 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking
4 The tool then moves in a semicircle to the hole center

Thread milling

5 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread
6 The tool then approaches the nominal thread diameter tangentially in a helical movement
7 The tool moves on a continuous helical downward path until the thread depth value is reached
8 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
9 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to the 2nd setup clearance

Please note while programming:

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

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

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

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

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

NOTICE

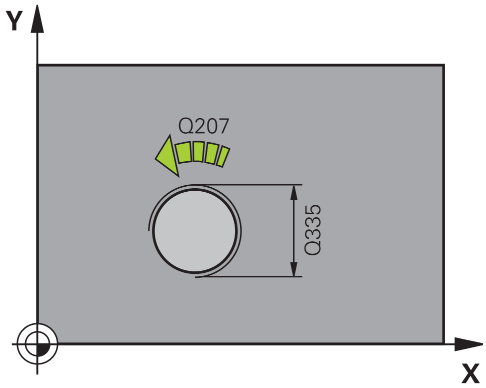
Danger of collision!

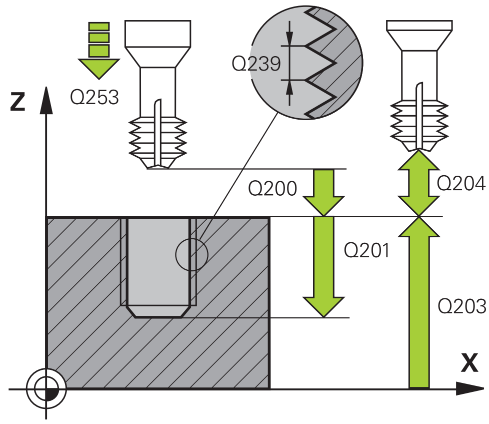
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

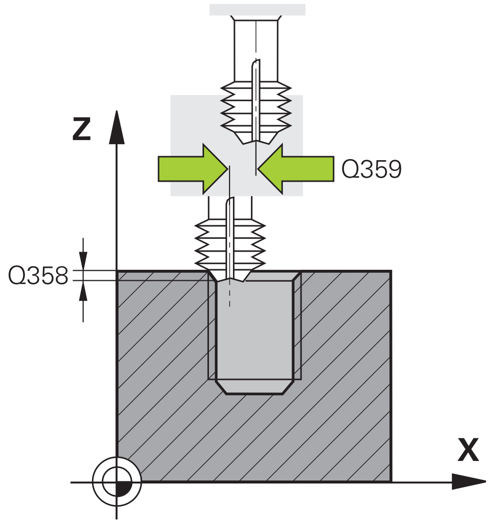
Cycle parameters

- **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  –= left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range 0 to 99999.9999
- **Q360 Countersink (before/after:0/1)?** : Machining the chamfer
  0 = before thread milling
  1 = after thread milling
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999





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



Example

25 CYCL DEF 265 HEL. THREAD DRLG/ MLG	
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;THREAD PITCH
Q201=-16	;DEPTH OF THREAD
Q253=750	;F PRE-POSITIONING
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q360=0	;COUNTERSINK PROCESS
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLNG

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

Cycle run

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

Countersinking at front

2 The control approaches the starting point for countersinking at front, starting from the center of the stud, on the reference axis in the working plane. The position of the starting point is determined by the thread radius, tool radius and pitch
3 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
4 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking
5 The tool then moves on a semicircle to the starting point

Thread milling

6 The control positions the tool at the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front
7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
8 The tool then approaches the nominal thread diameter tangentially in a helical movement
9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
10 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
11 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to the 2nd setup clearance

Please note while programming:

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

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

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

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

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

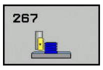
NOTICE

Danger of collision!

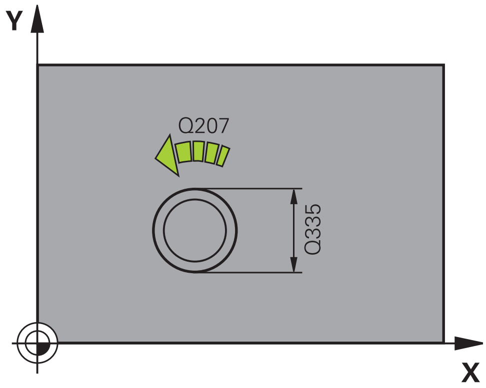
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

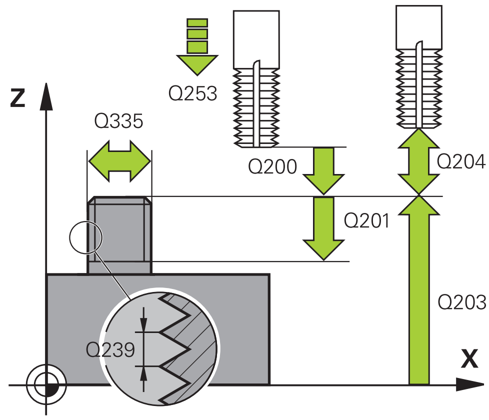
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters

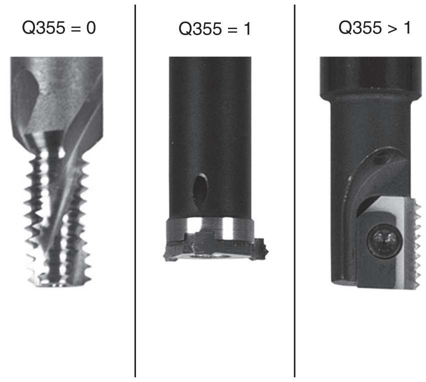


- **Q335 Nominal diameter?**: Nominal thread diameter. Input range 0 to 99999.9999
- **Q239 Pitch?**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  –= left-hand thread
  Input range: -99.9999 to +99.9999
- **Q201 Depth of thread?** (incremental): Distance between workpiece surface and bottom of the thread. Input range -99999.9999 to 99999.9999
- **Q355 Number of threads per step?**: Number of turns by which the tool is displaced:
  0 = one helix over the entire thread depth
  1 = continuous helix over the entire thread length
  >1 = several helix paths with approach and departure, between these, the control offsets the tool by **Q355** x pitch. Input range 0 to 99999
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3
  +1 = Climb
  -1 = Up-cut (if you enter 0, climb milling is performed)
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999





- **Q358 Sinking depth at front?** (incremental): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. Input range -99999.9999 to 99999.9999
- **Q359 Countersinking offset at front?** (incremental): Distance by which the control moves the tool center away from the center. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q254 Feed rate for counterboring?**: Traversing speed of the tool in mm/min during counterboring. Input range 0 to 99999.9999 alternatively **FAUTO**, **FU**
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**
- **Q512 Feed rate for approaching?**: Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input range 0 to 99999.999 alternatively **FAUTO**



Example

```
25 CYCL DEF 267  OUTSIDE THREAD
   MLLNG
   Q335=10      ;NOMINAL DIAMETER
   Q239=+1.5    ;THREAD PITCH
   Q201=-20     ;DEPTH OF THREAD
   Q355=0       ;THREADS PER STEP
   Q253=750     ;F PRE-POSITIONING
   Q351=+1      ;CLIMB OR UP-CUT
   Q200=2       ;SET-UP CLEARANCE
   Q358=+0      ;DEPTH AT FRONT
   Q359=+0      ;OFFSET AT FRONT
   Q203=+30     ;SURFACE COORDINATE
   Q204=50      ;2ND SET-UP CLEARANCE
   Q254=150     ;F COUNTERBORING
   Q207=500     ;FEED RATE FOR MILLNG
   Q512=0       ;FEED FOR APPROACH
```

5.11 Programming Examples

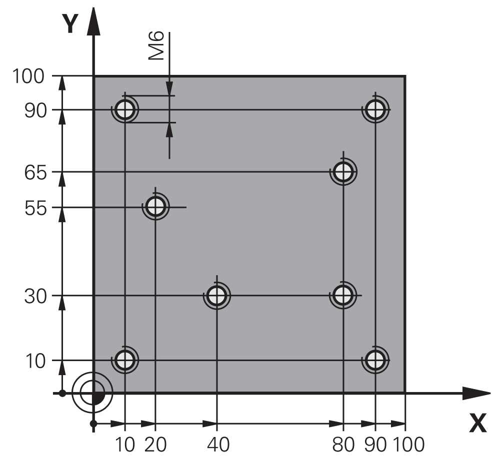
Example: Thread milling

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

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

Program sequence

- Centering
- Drilling
- Tapping



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

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

TAB1. PNT point table

TAB1. PNTMM
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]

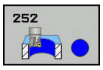
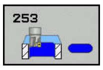
6

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	165
	252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	171
	253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal plunging	178
	254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal plunging	183
	256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	189
	257 CIRCULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required	194
	233 FACE MILLING Machining the face with up to 3 limits	204

6.2 RECTANGULAR POCKET (Cycle 251, ISO: G251)

Cycle run

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

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

Roughing

1 The tool plunges the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter Q366.
2 The control roughs out the pocket from the inside out, taking the path overlap (parameter Q370) and the finishing allowance (parameters Q368 and Q369) into account.
3 At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall, then moves to the set-up clearance above the current pecking depth and returns from there at rapid traverse to the pocket center.
4 This process is repeated until the programmed pocket depth is reached.

Finishing

5 If finishing allowances have been defined, the control plunges and then approaches the contour. The approach movement occurs on a radius in order to ensure a gentle approach. The control first finishes the pocket walls, with multiple infeeds, if so specified.
6 Then the control finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially

Please note while programming!

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

Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.

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

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

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

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

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

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

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

NOTICE

Danger of collision!

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

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

NOTICE

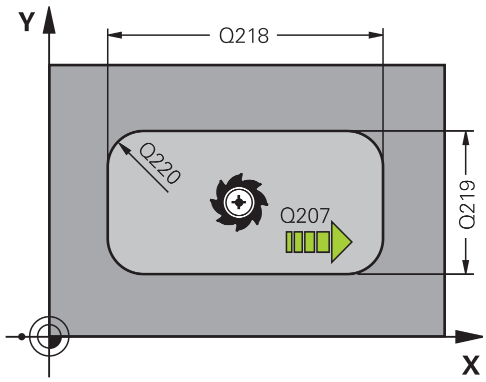
Danger of collision!

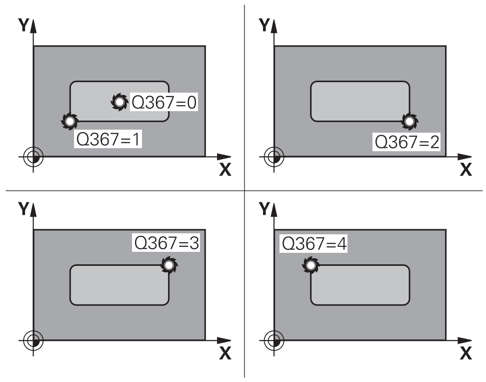
If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

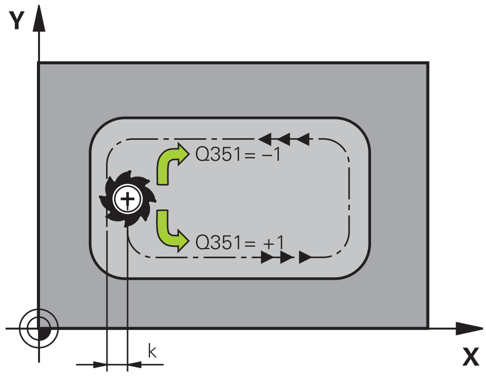
- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

Cycle parameters

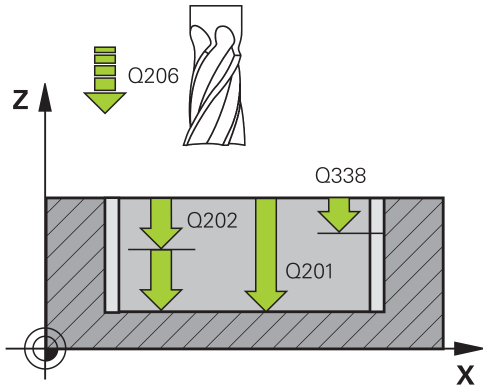
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q218 First side length?** (incremental): Pocket length, parallel to the reference axis of the working plane Input range: 0 to 99999.9999
- **Q219 Second side length?** (incremental): Pocket length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q220 Corner radius?**: Radius of the pocket corner. If you have entered 0 here, the control assumes that the corner radius is equal to the tool radius. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Q224 Angle of rotation?** (absolute): Angle by which the entire machining configuration is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: -360.0000 to 360.0000
- **Q367 Position of pocket (0/1/2/3/4)?**: Position of the pocket with respect to the position of the tool when the cycle is called:
  0: Tool position = pocket center
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999

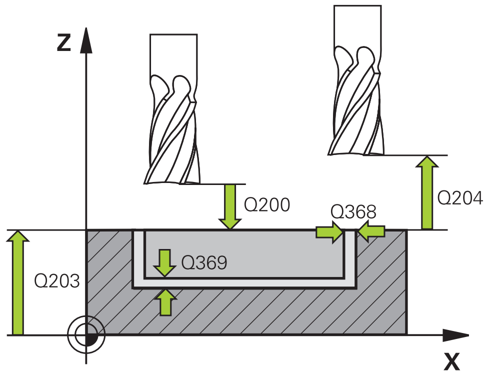






- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1,9999; alternatively **PREDEF**





Example

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

- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
  0: vertical plunging. The control plunges the tool perpendicularly, regardless of the **ANGLE** plunging angle defined in the tool table
  1: helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control generates an error message
  2: reciprocating plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. The reciprocation length depends on the plunging angle. As a minimum value, the control uses twice the tool diameter
  PREFEF: The control uses the value from the GLOBAL DEF block
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**
- **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
  0: Feed rate with respect to the tool center point path
  1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
  2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
  3: Feed rate always refers to the cutting edge

```
  Q207=500   ;FEED RATE FOR MILLNG
  Q351=+1    ;CLIMB OR UP-CUT
  Q201=-20   ;DEPTH
  Q202=5     ;PLUNGING DEPTH
  Q369=0.1   ;ALLOWANCE FOR FLOOR
  Q206=150   ;FEED RATE FOR PLNGNG
  Q338=5     ;INFEED FOR FINISHING
  Q200=2     ;SET-UP CLEARANCE
  Q203=+0    ;SURFACE COORDINATE
  Q204=50    ;2ND SET-UP CLEARANCE
  Q370=1     ;TOOL PATH OVERLAP
  Q366=1     ;PLUNGE
  Q385=500   ;FINISHING FEED RATE
  Q439=0     ;FEED RATE REFERENCE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

6.3 CIRCULAR POCKET (Cycle 252, ISO: G252)

Cycle run

Use Cycle 252 CIRCULAR POCKET to machine circular pockets. Depending on the cycle parameters, the following machining alternatives are available:

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

Roughing

1. The control first moves the tool at rapid traverse to the set-up clearance Q200 above the workpiece
2. The tool plunges to the first plunging depth at the pocket center. Specify the plunging strategy with parameter Q366.
3. The control roughs out the pocket from the inside out, taking the path overlap (parameter Q370) and the finishing allowance (parameters Q368 and Q369) into account.
4. At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall to the set-up clearance Q200 in the working plane, then retracts the tool by Q200 at rapid traverse and returns it from there at rapid traverse to the pocket center
5. Steps 2 to 4 are repeated until the programmed pocket depth is reached, taking the finishing allowance Q369 into account.
6. If only roughing was programmed (Q215=1) the tool moves away from the pocket wall tangentially by the set-up clearance Q200, then retracts at rapid traverse to the 2nd set-up clearance Q204 in the tool axis and returns at rapid traverse to the pocket center

Finishing

1 If finishing allowances have been defined, the control first finishes the pocket walls, in multiple infeeds, if so specified.
2 The control positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance Q368 and the set-up clearance Q200
3 The control roughs out the pocket from the inside out, until the diameter Q223 is reached
4 Then, the control positions the tool again in the tool axis near the pocket wall at a distance corresponding to the finishing allowance Q368 and the set-up clearance Q200 and repeats the finishing procedure for the side wall at the new depth
5 The control repeats this process until the programmed diameter is reached
6 After machining to the diameter Q223, the control retracts the tool tangentially by the finishing allowance Q368 plus the set-up clearance Q200 in the working plane, then retracts it at rapid traverse to the set-up clearance Q200 in the tool axis and returns it to the pocket center.
7 Next, the control moves the tool in the tool axis to the depth Q201 and finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially.
8 The control repeats this process until the depth Q201 plus Q369 is reached.
9 Finally, the tool moves away from the pocket wall tangentially by the set-up clearance Q200, then retracts at rapid traverse to the set-up clearance Q200 in the tool axis and returns at rapid traverse to the pocket center.

Please note while programming:

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

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

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

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

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

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

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

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

NOTICE

Danger of collision!

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

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

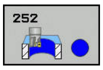
NOTICE

Danger of collision!

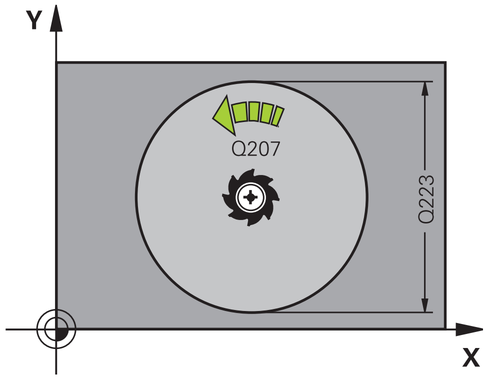
If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

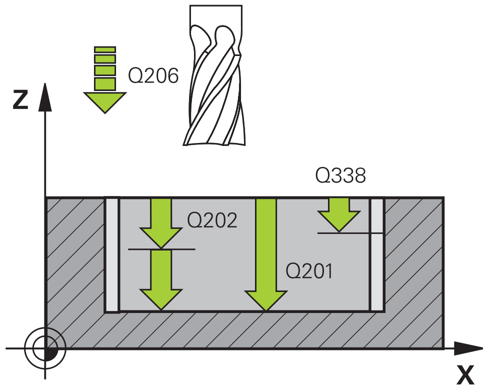
- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

Cycle parameters

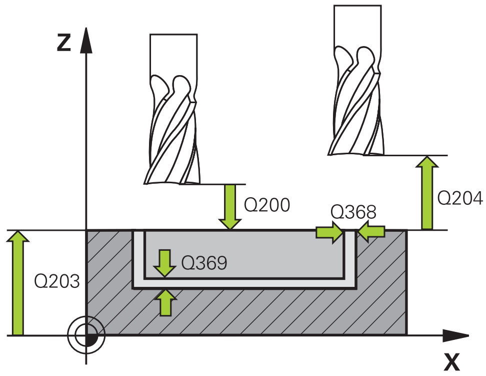


- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q223 Circle diameter?**: Diameter of the finished pocket. Input range 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**





- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input range: 0.1 to 1.9999; alternatively **PREDEF**
- **Q366 Plunging strategy (0/1)?**: Type of plunging strategy:
  - 0 = vertical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as 0 or 90. Otherwise, the control will display an error message
  - 1 = Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message
  - Alternative: **PREDEF**



Example

8 CYCL DEF 252 CIRCULAR POCKET	
Q215=0	**;MACHINING OPERATION**
Q223=60	**;CIRCLE DIAMETER**
Q368=0.2	**;ALLOWANCE FOR SIDE**
Q207=500	**;FEED RATE FOR MILLNG**
Q351=+1	**;CLIMB OR UP-CUT**
Q201=-20	**;DEPTH**
Q202=5	**;PLUNGING DEPTH**
Q369=0.1	**;ALLOWANCE FOR FLOOR**
Q206=150	**;FEED RATE FOR PLNGNG**
Q338=5	**;INFEED FOR FINISHING**
Q200=2	**;SET-UP CLEARANCE**
Q203=+0	**;SURFACE COORDINATE**
Q204=50	**;2ND SET-UP CLEARANCE**
Q370=1	**;TOOL PATH OVERLAP**
Q366=1	**;PLUNGE**
Q385=500	**;FINISHING FEED RATE**
Q439=3	**;FEED RATE REFERENCE**
9 L X+50 Y+50 R0 FMAX M3 M99	

- **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)

Cycle run

Use Cycle 253 to completely machine a slot. Depending on the cycle parameters, the following machining alternatives are available:

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

Roughing

1. Starting from the left slot arc center, the tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter Q366.
2. The control roughs out the slot from the inside out, taking the finishing allowances (parameters Q368 and Q369) into account
3. The control retracts the tool to the set-up clearance Q200. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed
4. This process is repeated until the programmed slot depth is reached.

Finishing

5. If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially in the left slot arc
6. Then the control finishes the floor of the slot from the inside out.

Please note while programming:

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

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

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

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

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

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

NOTICE

Danger of collision!

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

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

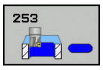
NOTICE

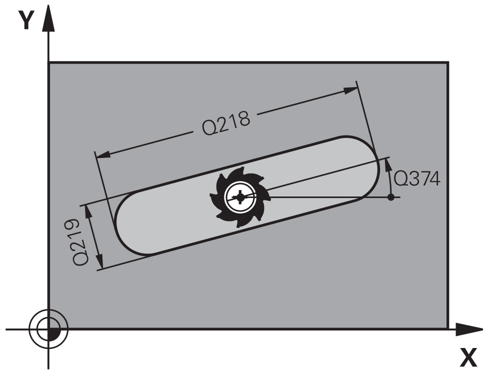
Danger of collision!

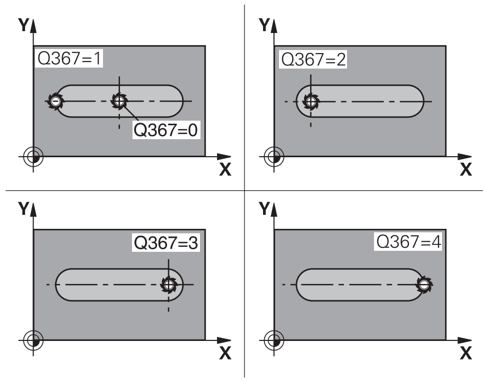
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

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

Cycle parameters

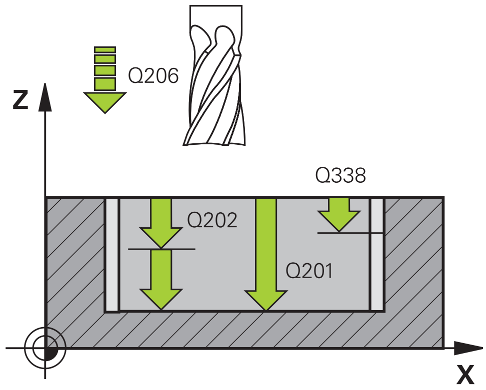


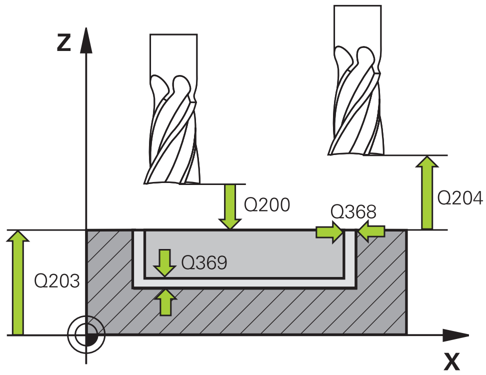




- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q218 Length of slot?** (value parallel to the reference axis of the working plane): Enter the length of the slot. Input range: 0 to 99999.9999
- **Q219 Width of slot?** (value parallel to the minor axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Q374 Angle of rotation?** (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: -360.000 to 360.000
- **Q367 Position of slot (0/1/2/3/4)?**: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position = slot center
  1: Tool position = left end of slot
  2: Tool position = center of left slot arc
  3: Tool position = center of right slot arc
  4: Tool position = right end of slot

- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: -99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999





Example

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

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
  - 0 = vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated.
  - 1, 2 = reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message
  - Alternative: **PREDEF**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**
- **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
  0: Feed rate with respect to the tool center point path
  1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
  2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
  3: Feed rate always refers to the cutting edge

```
    Q351=+1     ;CLIMB OR UP-CUT
    Q201=-20    ;DEPTH
    Q202=5      ;PLUNGING DEPTH
    Q369=0.1    ;ALLOWANCE FOR FLOOR
    Q206=150    ;FEED RATE FOR PLNGNG
    Q338=5      ;INFEED FOR FINISHING
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q366=1      ;PLUNGE
    Q385=500    ;FINISHING FEED RATE
    Q439=0      ;FEED RATE REFERENCE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

6.5 CIRCULAR SLOT (Cycle 254, ISO: G254)

Cycle run

Use Cycle 254 to completely machine a circular slot. Depending on the cycle parameters, the following machining alternatives are available:

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

Roughing

1. The tool moves in a reciprocating motion in the slot center at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter Q366.
2. The control roughs out the slot from the inside out, taking the finishing allowances (parameters Q368 and Q369) into account
3. The control retracts the tool to the set-up clearance Q200. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed
4. This process is repeated until the programmed slot depth is reached.

Finishing

5. If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially
6. Then the control finishes the floor of the slot from the inside out

Please note while programming:

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

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

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

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

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

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

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

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

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

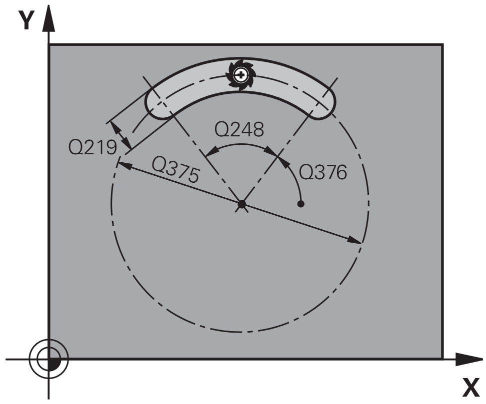
NOTICE

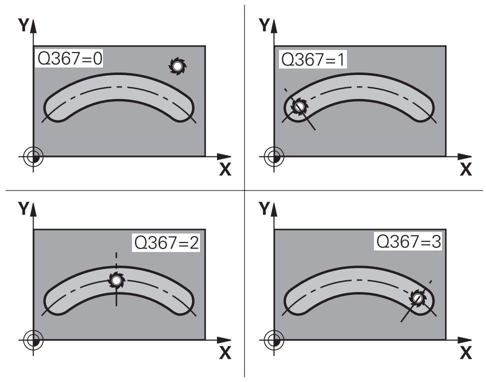
Danger of collision!

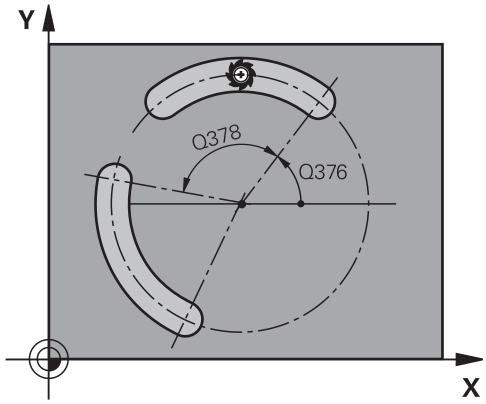
If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

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

Cycle parameters

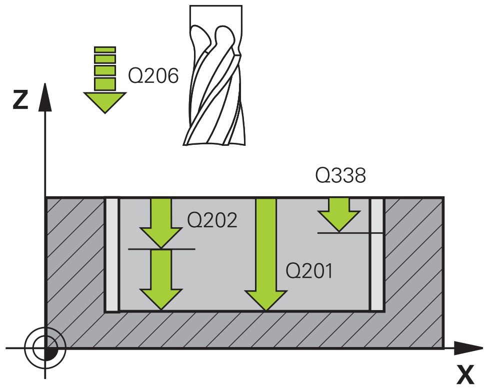


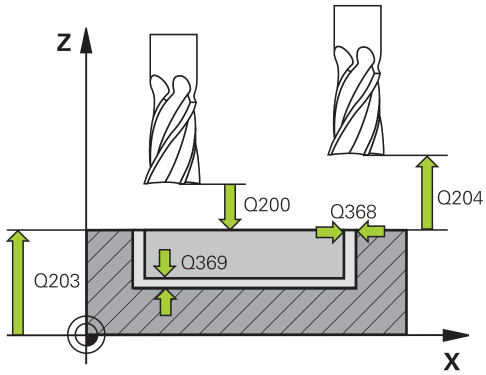




- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q219 Width of slot?** (value parallel to the minor axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Q375 Pitch circle diameter?**: Enter the diameter of the pitch circle. Input range: 0 to 99999.9999
- **Q367 Ref. for slot pos. (0/1/2/3)?**: Position of the slot in reference to the position of the tool when the cycle is called:
  0: Tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle
  1: Tool position = center of left slot arc. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account
  2: Tool position = center of centerline. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account
  3: Tool position = center of right slot arc. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.
- **Q216 Center in 1st axis?** (absolute): Center of the pitch circle in the reference axis of the working plane. **Only effective if Q367 = 0**. Input range: -99999.9999 to 99999.9999

- **Q217 Center in 2nd axis?** (absolute): Center of the pitch circle in the minor axis of the working plane. **Only effective if Q367 = 0**. Input range: -99999.9999 to 99999.9999
- **Q376 Starting angle?** (absolute): Enter the polar angle of the starting point. Input range: -360.000 to 360.000
- **Q248 Angular length?** (incremental): Enter the angle between the starting point and the end point of the slot. Input range: 0 to 360.000
- **Q378 Intermediate stepping angle?** (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. Input range: -360.000 to 360.000
- **Q377 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range: 1 to 99999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: -99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**





Example

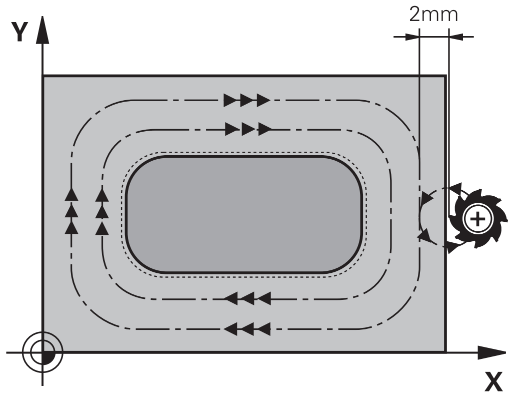
8 CYCL DEF 254 CIRCULAR SLOT	
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q375=80	;PITCH CIRCLE DIAMETR
Q367=0	;REF. SLOT POSITION
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q376=+45	;STARTING ANGLE
Q248=90	;ANGULAR LENGTH
Q378=0	;STEPPING ANGLE
Q377=1	;NR OF REPETITIONS

- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
  0: vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated.
  1, 2: reciprocal plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control generates an error message
  PREDEF: The control uses the value from the GLOBAL DEF block
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**
- **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
  0: Feed rate with respect to the tool center point path
  1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
  2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
  3: Feed rate always refers to the cutting edge

Q207=500	;FEED RATE FOR MILLNG
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=1	;PLUNGE
Q385=500	;FINISHING FEED RATE
Q439=0	;FEED RATE REFERENCE
9 L X+50 Y+50 R0 FMAX M3 M99	

6.6 RECTANGULAR STUD (Cycle 256, ISO: G256)

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:

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

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

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

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

NOTICE

Danger of collision!

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

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

NOTICE

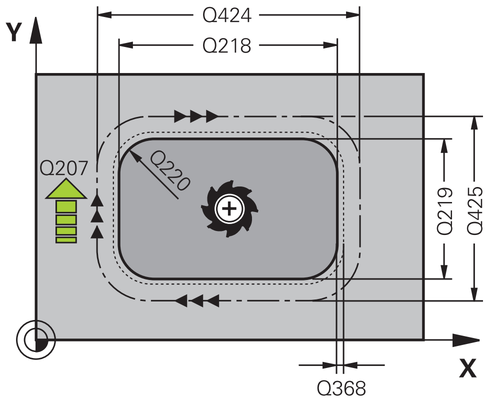
Danger of collision!

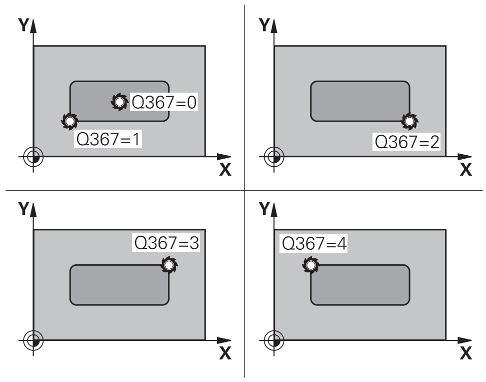
If there is not enough room for the approach movement next to the stud, there is danger of collision.

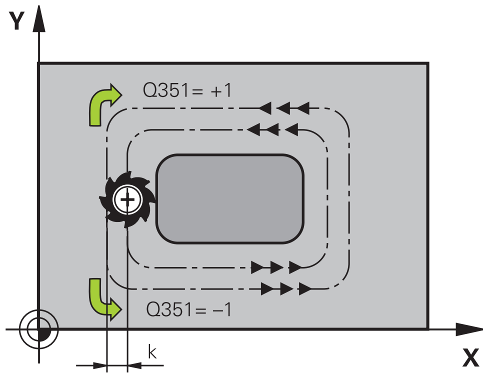
- Depending on the approach position Q439, leave enough room next to the stud for the approach movement
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position.

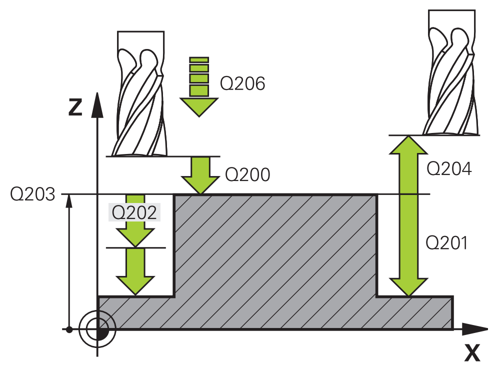
Cycle parameters

- **Q218 First side length?**: Stud length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q424 Workpiece blank side length 1?**: Length of the stud blank, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **First side length**. The control performs multiple lateral stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q219 Second side length?**: Stud length, parallel to the minor axis of the working plane. Enter **Workpiece blank side length 2** greater than **Second side length**. The control performs multiple lateral stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q425 Workpiece blank side length 2?**: Length of the stud blank, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q220 Radius / Chamfer (+/-)?**: Enter the value for the radius or chamfer form element. If you enter a positive value between 0 and +99999.9999, the control rounds every corner. The value you enter here refers to the radius. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane, is left over after machining. Input range: 0 to 99999.9999
- **Q224 Angle of rotation?** (absolute): Angle by which the entire machining configuration is rotated. The center of rotation is the position at which the tool is located when the cycle is called. Input range: -360.0000 to 360.0000
- **Q367 Position of stud (0/1/2/3/4)?**: Position of the stud in reference to the position of the tool when the cycle is called:
  0: Tool position = stud center
  1: Tool position = lower left corner
  2: Tool position = lower right corner
  3: Tool position = upper right corner
  4: Tool position = upper left corner









- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX** , **FAUTO**, **FU** , **FZ**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input range: 0.1 to 1.9999; alternatively **PREDEF**

Example

8 CYCL DEF 256 RECTANGULAR STUD	
Q218=60	;FIRST SIDE LENGTH
Q424=74	;WORKPC. BLANK SIDE 1
Q219=40	;2ND SIDE LENGTH
Q425=60	;WORKPC. BLANK SIDE 2
Q220=5	;CORNER RADIUS
Q368=0.2	;ALLOWANCE FOR SIDE
Q224=+0	;ANGLE OF ROTATION
Q367=0	;STUD POSITION
Q207=500	;FEED RATE FOR MILLNG
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q370=1	;TOOL PATH OVERLAP
Q437=0	;APPROACH POSITION
Q215=1	;MACHINING OPERATION
Q369=+0	;ALLOWANCE FOR FLOOR
Q338=+0	;INFEED FOR FINISHING
Q385=+0	;FEED RATE FOR FINISHING
9 L X+50 Y+50 R0 FMAX M3 M99	

- **Q437 Starting position (0...4)?**: Define the approach strategy of the tool:
  0: Right of the stud (default setting)
  1: lower left corner
  2: lower right corner
  3: upper right corner
  4: upper left corner.
  If approach marks should be appear on the stud surface during approach with the setting Q437=0, then choose another approach position.
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

6.7 CIRCULAR STUD (Cycle 257, ISO: G257)

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:

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

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

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

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

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

NOTICE

Danger of collision!

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

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

NOTICE

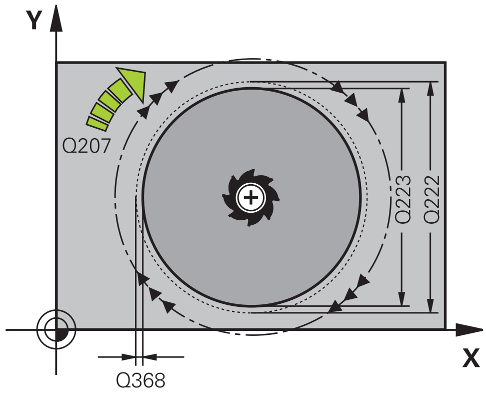
Danger of collision!

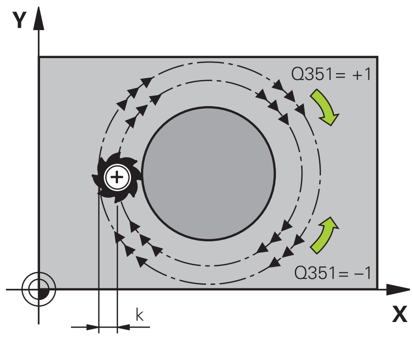
There is a danger of collision if there is insufficient room next to the stud.

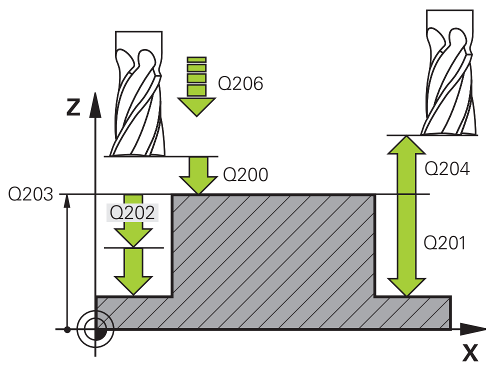
- In this cycle, the control performs an approach movement
- To define the precise starting position, enter a starting angle of 0° to 360° in parameter Q376
- Depending on the starting angle Q376, the following amount of space must be left next to the stud: At least tool diameter +2 mm
- If you use the default value -1 the control automatically calculates the starting position

Cycle parameters

- **Q223 Finished part diameter?**: Diameter of the completely machined stud. Input range: 0 to 99999.9999
- **Q222 Workpiece blank diameter?**: Diameter of the workpiece blank. The workpiece blank diameter must be greater than the diameter of the finished part. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX** , **FAUTO**, **FU** , **FZ**







- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1,9999; alternatively **PREDEF**
- **Q376 Starting angle?**: Polar angle relative to the stud center from where the tool approaches the stud. Input range: 0 to 359°
- **Q215 Machining operation (0/1/2)?**: Define the scope of machining:
  0: Roughing and finishing
  1: Roughing only
  2: Finishing only
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

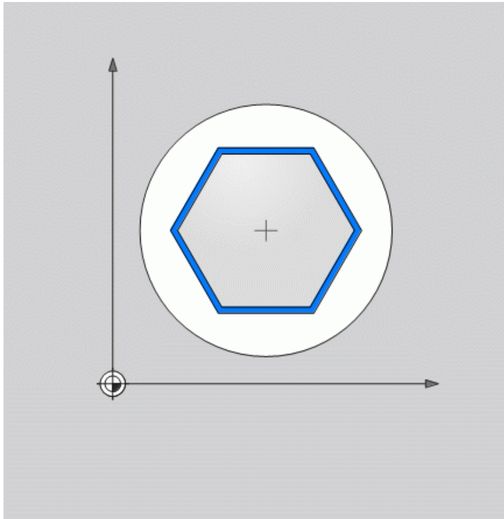
Example

```
8 CYCL DEF 257  CIRCULAR STUD
    Q223=60     ;FINISHED PART DIA.
    Q222=60     ;WORKPIECE BLANK DIA.
    Q368=0.2    ;ALLOWANCE FOR SIDE
    Q207=500    ;FEED RATE FOR MILLNG
    Q351=+1     ;CLIMB OR UP-CUT
    Q201=-20    ;DEPTH
    Q202=5      ;PLUNGING DEPTH
    Q206=150    ;FEED RATE FOR PLNGNG
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q370=1      ;TOOL PATH OVERLAP
    Q376=0      ;STARTING ANGLE
    Q215=+1     ;MACHINING OPERATION
    Q369=0      ;ALLOWANCE FOR FLOOR
    Q338=0      ;INFEED FOR FINISHING
    Q385=+500 ;FINISHING FEED RATE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

6.8 POLYGON STUD (Cycle 258, ISO: G258)

Cycle run

With the cycle **Polygon stud** you can create an even polygon by machining the contour outside. The milling operation is carried out on a spiral path, based on the diameter of the workpiece blank.



1 If the tool is below the 2nd set-up clearance at the beginning of machining, the control retracts the tool to the 2nd set-up clearance
2 Starting from the center of the stud the control moves the tool to the starting point of stud machining. The starting point depends, among others, on the diameter of the workpiece blank and the angle of rotation of the stud. The angle of rotation is determined with parameter Q224
3 The tool moves at rapid traverse **FMAX** to the setup clearance Q200 and from there with the feed rate for plunging to the first plunging depth.
4 The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5 The control moves the tool on a tangential path from the outside to the inside
6 The tool will be lifted in the direction of the spindle axis to the 2nd setup clearance in one rapid movement
7 If several plunging depths are required, the control returns the tool to the starting point of the stud milling process and then plunges the tool to the programmed depth
8 This process is repeated until the programmed stud depth is reached.
9 At the end of the cycle, first a departing motion is performed. Then the control will move the tool on the tool axis to the 2nd set-up clearance

Please note while programming:

Before the start of the cycle you will have to preposition the tool on the machining plane. In order to do so, move the tool with radius compensation **R0** to the center of the stud.

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

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

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

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

In this cycle, the control performs an automatic approach movement. If there is not enough space, a collision might occur.

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

NOTICE

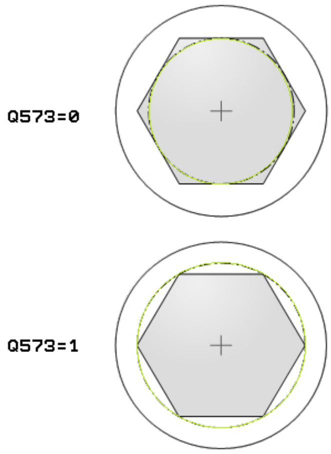
Danger of collision!

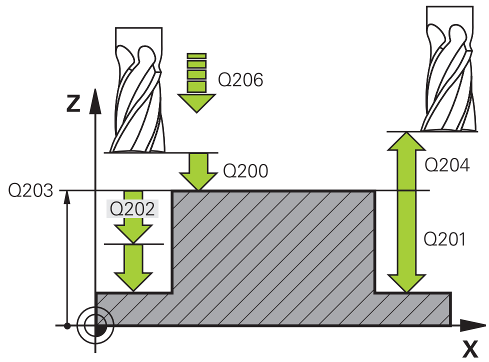
At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle need not be the same as the starting position.

- Control the traversing movements of the machine
- In the simulation, control the end position of the tool after the cycle
- After the cycle, program the absolute (not incremental) coordinates

Cycle parameters

- **Q573 Inscr.circle/circumcircle (0/1)?**: Definition of whether the dimensioning shall reference the inscribed circle or the perimeter:
  0= dimensioning refers to the inscribed circle
  1= dimensioning refers to the perimeter
- **Q571 Reference circle diameter?**: Definition of the diameter of the reference circle. Specify in parameter Q573 whether the diameter references the inscribed circle or the perimeter. Input range: 0 to 99999.9999
- **Q222 Workpiece blank diameter?**: Definition of the diameter of the workpiece blank. The workpiece blank diameter must be greater than the reference circle diameter. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input range: 0 to 99999.9999
- **Q572 Number of corners?**: Enter the number of corners of the polygon stud. The control distributes the corners evenly on the stud. Input range: 3 to 30
- **Q224 Angle of rotation?**: Specify which angle is used to machine the first corner of the polygon stud. Input range: -360° to +360°





- **Q220 Radius / Chamfer (+/-)?**: Enter the value for the radius or chamfer form element. If you enter a positive value between 0 and +99999.9999, the control rounds every corner. The value you enter here refers to the radius. If you enter a negative value between 0 and -99999.9999 all corners of the contour are chamfered and the value entered refers to the length of the chamfer.
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. If you enter a negative value here, the control will return the tool to a diameter outside of the workpiece blank diameter after roughing. Input range -99999.9999 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of stud. Input range: -99999.9999 to 99999.9999
- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**, **FU**, **FZ**

Example

```
8 CYCL DEF 258 POLYGON STUD
    Q573=1      ;REFERENCE CIRCLE
    Q571=50     ;REF-CIRCLE DIAMETER
    Q222=120    ;WORKPIECE BLANK DIA.
    Q572=10     ;NUMBER OF CORNERS
    Q224=40     ;ANGLE OF ROTATION
    Q220=2      ;RADIUS / CHAMFER
    Q368=0      ;ALLOWANCE FOR SIDE
    Q207=3000   ;FEED RATE FOR MILLNG
    Q351=1      ;CLIMB OR UP-CUT
    Q201=-18    ;DEPTH
    Q202=10     ;PLUNGING DEPTH
    Q206=150    ;FEED RATE FOR PLNGNG
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q370=1      ;TOOL PATH OVERLAP
    Q215=0      ;MACHINING OPERATION
    Q369=0      ;ALLOWANCE FOR FLOOR
    Q338=0      ;INFEED FOR FINISHING
    Q385=500    ;FINISHING FEED RATE
9 L  X+50 Y+50  R0 FMAX  M3 M99
```

- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q370 Path overlap factor?**: Q370 x tool radius = stepover factor k. Input range: 0.0001 to 1,9999; alternatively **PREDEF**
- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**

6.9 FACE MILLING (Cycle 233, ISO: G233)

Cycle run

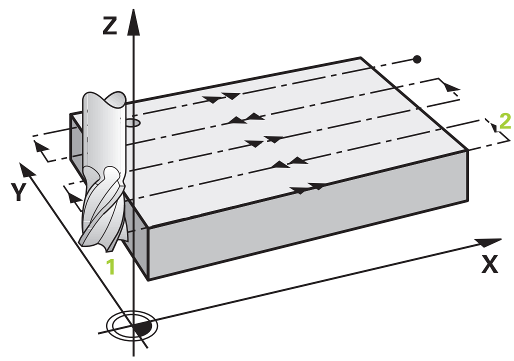
With Cycle 233, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

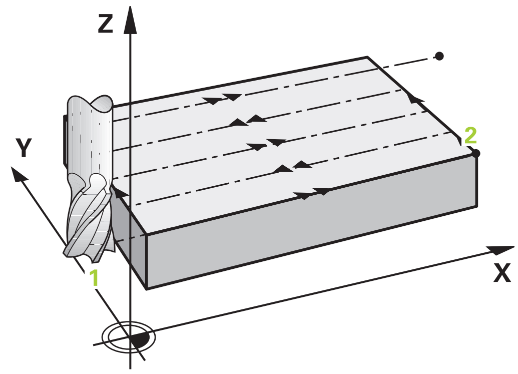
- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover when retracting at rapid traverse
-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

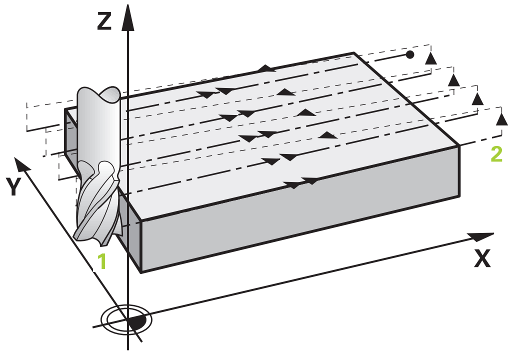


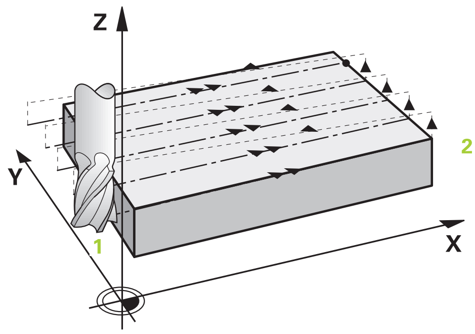


4 The control moves the tool to end point **2** at the programmed feed rate for milling
5 Then the control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius, the maximum path overlap factor and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

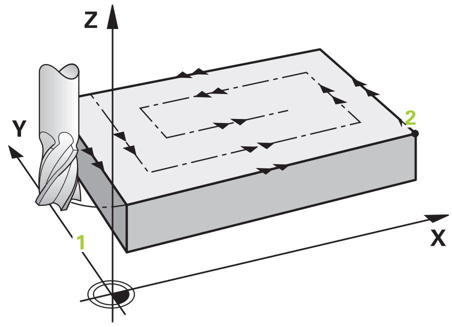




4 The tool subsequently moves to end point **2** at the programmed feed rate for milling
5 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves at **FMAX**directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side.
6 The tool then returns to the current infeed depth and moves in the direction of end point **2**
7 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**
8 If more than one infeed is required, the control moves the tool in the tool axis to the next plunging depth at the positioning feed rate
9 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate
10 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance

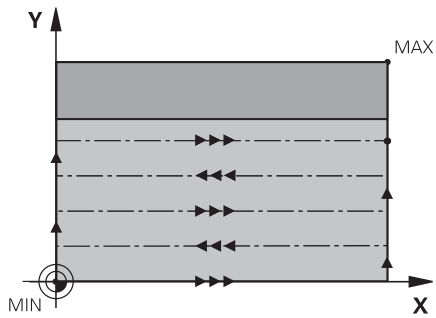
Strategy Q389=4

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:

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Keep in mind the machining direction.

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

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

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

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

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

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

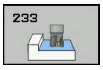
NOTICE

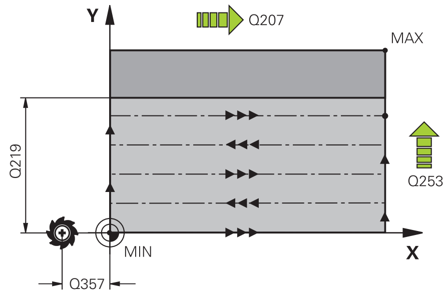
Danger of collision!

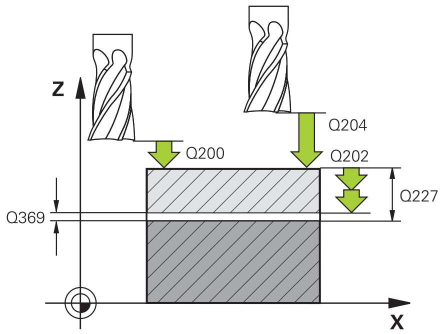
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

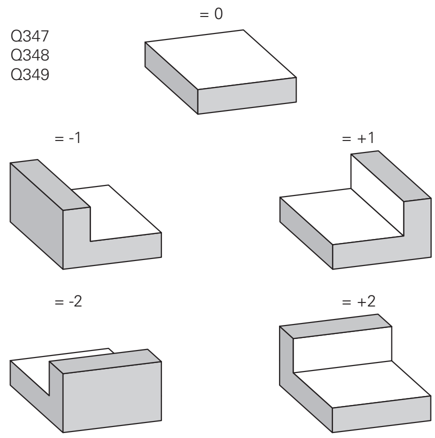
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

Cycle parameters









- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q389 Machining strategy (0-4)?**: Determine how the control should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface being machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate outside the surface to be machined
  3: Line-by-line machining, retraction and stepover at positioning feed rate at the edge of the surface to be machined
  4: Helical machining, uniform infeed from the outside toward the inside
- **Q350 Milling direction?**: Axis in the working plane that defines the machining direction:
  1: Reference axis = machining direction
  2: Minor axis = machining direction
- **Q218 First side length?** (incremental): Length of the surface to be machined in the reference axis of the working plane, referencing the starting point in the 1st axis. Input range -99999.9999 to 99999.9999
- **Q219 Second side length?** (incremental): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **STARTNG PNT 2ND AXIS**. Input range: -99999.9999 to 99999.9999

- **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range: -99999.9999 to 99999.9999
- **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis to which the surface is to be face-milled. Input range: -99999.9999 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range: 0 to 99999.9999
- **Q202 MAX. PLUNGING DEPTH** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q370 Path overlap factor?**: Maximum stepover factor k. The control calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. Input range: 0.1 to 1.9999.
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (Q389=1), the control uses the cross feed rate for milling Q207. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- **Q357 Safety clearance to the side?** (incremental) Parameter Q357 affects the following situations:
  Approaching the first plunging depth: Q357 is the lateral distance between tool and workpiece
  Roughing with the Q389=0-3 roughing strategies: The surface to be machined is extended in **Q350** MILLING DIRECTION by the value from Q357 if no limit has been set in that direction
  Side finishing: The paths will be extended by Q357 in **Q350** MILLING DIRECTION.
  Input range: 0 to 99999.9999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**

Example

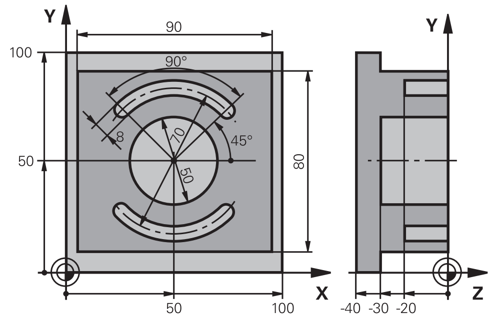
8 CYCL DEF 233 FACE MILLING	
Q215=0	;MACHINING OPERATION
Q389=2	;MILLING STRATEGY
Q350=1	;MILLING DIRECTION
Q218=120	;FIRST SIDE LENGTH
Q219=80	;2ND SIDE LENGTH
Q227=0	;STARTNG PNT 3RD AXIS
Q386=-6	;END POINT 3RD AXIS
Q369=0.2	;ALLOWANCE FOR FLOOR
Q202=3	;MAX. PLUNGING DEPTH
Q370=1	;TOOL PATH OVERLAP
Q207=500	;FEED RATE FOR MILLNG
Q385=500	;FINISHING FEED RATE
Q253=750	;F PRE-POSITIONING
Q357=2	;CLEARANCE TO SIDE
Q200=2	;SET-UP CLEARANCE
Q204=50	;2ND SET-UP CLEARANCE
Q347=0	;1ST LIMIT
Q348=0	;2ND LIMIT
Q349=0	;3RD LIMIT
Q220=2	;CORNER RADIUS
Q368=0	;ALLOWANCE FOR SIDE
Q338=0	;INFEED FOR FINISHING
Q367=-1	;SURFACE POSITION (-1/0/1/2/3/4)?
9 L X+0 Y+0 R0 FMAX M3 M99	

- **Q347 1st limit?**: Select the side of the workpiece where the level surface is bordered by a side wall (not possible with helical machining). Depending on the position of the side wall, the control limits machining of the level surface to the respective coordinate of the starting point or to the side length: (not possible with helical machining):
  Input **0**: No limiting
  Input **-1**: Limit in negative reference axis
  Input **+1**: Limiting in positive reference axis
  Input **-2**: Limiting in negative minor axis
  Input **+2**: Limiting in positive minor axis
- **Q348 2nd limit?**: See parameter 1st limit Q347
- **Q349 3rd limit?**: See parameter 1st limit Q347
- **Q220 Corner radius?**: Radius of a corner at limits (Q347 to Q349). Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999

- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q367 Surface position (-1/0/1/2/3/4)?**: Position of the surface referencing the position of the tool when the cycle is called:
  -1: Tool position = current position
  0: Tool position = stud center
  1: Tool position = Lower left corner
  2: Tool position = Lower right corner
  3: Tool position = Upper right corner
  4: Tool position = Upper left corner

6.10 Programming Examples

Example: Milling pockets, studs and slots



0 BEGINN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call: roughing/finishing
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD	Cycle definition: outside machining
Q218=90 ;FIRST SIDE LENGTH	
Q424=100 ;WORKPC. BLANK SIDE 1	
Q219=80 ;2ND SIDE LENGTH	
Q425=100 ;WORKPC. BLANK SIDE 2	
Q220=0 ;CORNER RADIUS	
Q368=0 ;ALLOWANCE FOR SIDE	
Q224=0 ;ANGLE OF ROTATION	
Q367=0 ;STUD POSITION	
Q207=250 ;FEED RATE FOR MILLNG	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q437=0 ;APPROACH POSITION	
6 L X+50 Y+50 R0 M3 M99	Cycle call for outside machining
7 CYCL DEF 252 CIRCULAR POCKET	Cycle definition: circular pocket
Q215=0 ;MACHINING OPERATION	
Q223=50 ;CIRCLE DIAMETER	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q207=500 ;FEED RATE FOR MILLNG	

Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q366=1 ;PLUNGE	
Q385=750 ;FINISHING FEED RATE	
Q439=0 ;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99	Cycle call for circular pocket
9 L Z+250 R0 FMAX M6	Retract the tool
10 TOOL CALL 2 Z S5000	Tool call: slot milling cutter
11 CYCL DEF 254 CIRCULAR SLOT	Cycle definition: slots
Q215=0 ;MACHINING OPERATION	
Q219=8 ;SLOT WIDTH	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q375=70 ;PITCH CIRCLE DIAMETR	
Q367=0 ;REF. SLOT POSITION	No pre-positioning in X/Y required
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q376=+45 ;STARTING ANGLE	
Q248=90 ;ANGULAR LENGTH	
Q378=180 ;STEPPING ANGLE	Starting point for second slot
Q377=2 ;NR OF REPETITIONS	
Q207=500 ;FEED RATE FOR MILLNG	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-20 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q366=1 ;PLUNGE	
Q385=500 ;FINISHING FEED RATE	
Q439=0 ;FEED RATE REFERENCE	
12 CYCL CALL FMAX M3	Cycle call for slots
13 L Z+250 R0 FMAX M2	Retract the tool, end program

14 END PGM C210 MM	

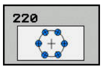
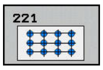
7

Fixed Cycles: Pattern Definitions

7.1 Fundamentals

Overview

The control provides two cycles for machining point patterns directly:

Soft key	Cycle	Page
	220 POLAR PATTERN	219
	221 CARTESIAN PATTERN	222

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

> If you have to machine irregular point patterns, use **CYCL CALL PAT** (see "Point tables", Page 75) to develop point tables.
>
> More regular point patterns are available with the **pattern def** function (see "Pattern definition with PATTERN DEF", Page 68).

Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL DRILLING
Cycle 204	BACK BORING
Cycle 205	UNIVERSAL PECKING
Cycle 206	TAPPING NEW with a floating tap holder
Cycle 207	RIGID TAPPING without a floating tap holder NEW
Cycle 208	BORE MILLING
Cycle 209	TAPPING WITH CHIP BREAKING
Cycle 240	CENTERING
Cycle 251	RECTANGULAR POCKET
Cycle 252	CIRCULAR POCKET MILLING
Cycle 253	SLOT MILLING
Cycle 254	CIRCULAR SLOT (can only be combined with Cycle 221)
Cycle 256	RECTANGULAR STUD
Cycle 257	CIRCULAR STUD
Cycle 262	THREAD MILLING
Cycle 263	THREAD MILLING/COUNTERSINKING
Cycle 264	THREAD DRILLING/MILLING
Cycle 265	HELICAL THREAD DRILLING/MILLING
Cycle 267	OUTSIDE THREAD MILLING

7.2 POLAR PATTERN (Cycle 220, ISO: G220)

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.

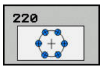
Please note while programming:

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

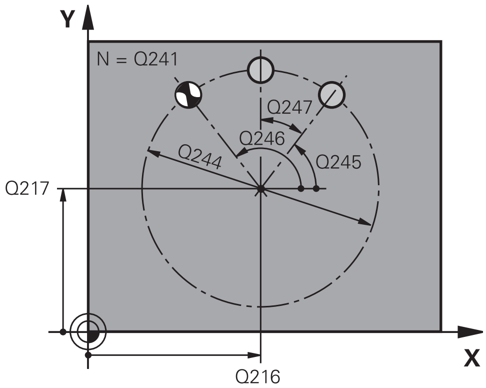
If you combine Cycle 220 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface and the 2nd set-up clearance that were defined in Cycle 220 or 221 will be effective. This applies within the NC program until the affected parameters are overwritten again. Example: If, in an NC program, Cycle 200 is defined with Q203=0 and you then program a Cycle 220 with Q203=-5, then the subsequent call with CYCL CALL and M99 will use Q203=-5. Cycles 220 and 221 overwrite the above-mentioned parameters of CALL-active machining cycles (if the same input parameters have been programmed in both cycles).

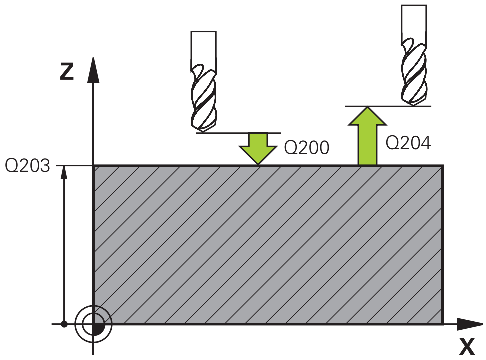
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q216 Center in 1st axis?** (absolute): Pitch circle center in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q217 Center in 2nd axis?** (absolute): Pitch circle center in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q244 Pitch circle diameter?**: Diameter of the pitch circle. Input range: 0 to 99999.9999
- **Q245 Starting angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range: -360.000 to 360.000
- **Q246 Stopping angle?** (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you specify a stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range: -360.000 to 360.000
- **Q247 Intermediate stepping angle?** (incremental): Angle between two machining operations on a pitch circle. If you enter a stepping angle of 0, the control will calculate the stepping angle from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the control will not take the stopping angle into account. The sign for the stepping angle determines the working direction (negative = clockwise). Input range: -360.000 to 360.000
- **Q241 Number of repetitions?**: Total number of machining positions on the pitch circle. Input range: 1 to 99999
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999





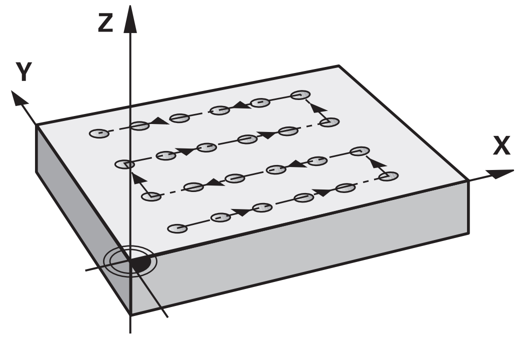
Example

```
53 CYCL DEF 220 POLAR PATTERN
    Q216=+50   ;CENTER IN 1ST AXIS
    Q217=+50   ;CENTER IN 2ND AXIS
    Q244=80    ;PITCH CIRCLE DIAMETR
    Q245=+0    ;STARTING ANGLE
    Q246=+360  ;STOPPING ANGLE
    Q247=+0    ;STEPPING ANGLE
    Q241=8     ;NR OF REPETITIONS
    Q200=2     ;SET-UP CLEARANCE
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q301=1     ;MOVE TO CLEARANCE
    Q365=0     ;TYPE OF TRAVERSE
```

- **Q301 Move to clearance height (0/1)?**: Definition of how the tool is to move between machining operations:
  0: Move to set-up clearance between machining operations
  1: Move to 2nd set-up clearance between machining operations
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool will move between machining operations:
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

7.3 LINEAR POINT PATTERN (Cycle 221, ISO: G221)

Cycle run



1 The control automatically moves the tool from its current position to the starting point for the first machining operation
   Sequence:
   - Move to the 2nd set-up clearance (spindle axis)
   - Approach the starting point in the machining plane
   - Move to the set-up clearance above the workpiece surface (spindle axis)
2 From this position, the control executes the last defined fixed machining cycle
3 Then, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis. The tool stops at the set-up clearance (or the 2nd set-up clearance)
4 This procedure (steps 1 to 3) will be repeated until all machining operations from the first line have been completed. The tool is located above the last point of the first line
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation
6 From this position, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis.
7 This process (6) is repeated until all machining operations in the second line have been executed.
8 The tool then moves to the starting point of the next line
9 All subsequent lines are processed in a reciprocating movement.

Please note while programming:

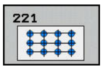
Cycle 221 is DEF active, which means that Cycle 221 automatically calls the last defined fixed cycle.

If you combine Cycle 221 with one of the fixed cycles 200 to 209 and 251 to 267, the set-up clearance, workpiece surface, the 2nd set-up clearance, and the rotational position that were defined in Cycle 221 will be effective.

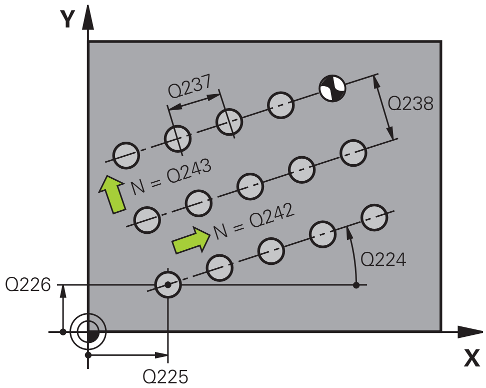
The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

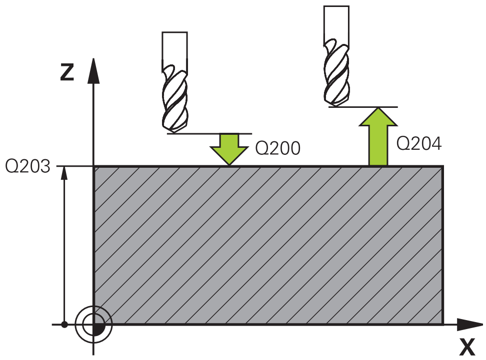
If you run this cycle in the Single Block mode of operation, the control stops between the individual points of a point pattern.

Cycle parameters



- **Q225 Starting point in 1st axis?** (absolute): Coordinate of the starting point in the reference axis of the working plane
- **Q226 Starting point in 2nd axis?** (absolute): Coordinate of the starting point in the minor axis of the working plane
- **Q237 Spacing in 1st axis?** (incremental): Spacing between the individual points on the line
- **Q238 Spacing in 2nd axis?** (incremental): Spacing between the individual lines
- **Q242 Number of columns?**: Number of machining operations on a line
- **Q243 Number of lines?**: Number of lines
- **Q224 Angle of rotation?** (absolute): Angle by which the entire pattern is rotated. The center of rotation is located at the starting point
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the tool is to move between machining operations:
  0: Move to set-up clearance between machining operations
  1: Move to 2nd set-up clearance between machining operations



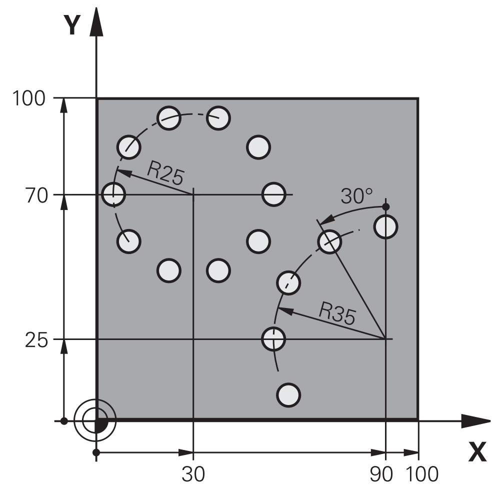


Example

54 CYCL DEF 221 CARTESIAN PATTERN	
Q225=+15	;STARTNG PNT 1ST AXIS
Q226=+15	;STARTNG PNT 2ND AXIS
Q237=+10	;SPACING IN 1ST AXIS
Q238=+8	;SPACING IN 2ND AXIS
Q242=6	;NUMBER OF COLUMNS
Q243=4	;NUMBER OF LINES
Q224=+15	;ANGLE OF ROTATION
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE

7.4 Programming Examples

Example: Polar hole patterns



0 BEGIN PGM HOLEPAT MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Tool call
4 L Z+250 R0 FMAX M3	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=4 ;PLUNGING DEPTH	
Q211=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=0 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
6 CYCL DEF 220 POLAR PATTERN	Define cycle for polar point pattern 1, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220
Q216=+30 ;CENTER IN 1ST AXIS	
Q217=+70 ;CENTER IN 2ND AXIS	
Q244=50 ;PITCH CIRCLE DIAMETR	
Q245=+0 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+0 ;STEPPING ANGLE	
Q241=10 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	

Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=0 ;TYPE OF TRAVERSE	
7 CYCL DEF 220 POLAR PATTERN	Define cycle for polar point pattern 2, CYCL 200 is called automatically; Q200, Q203 and Q204 are effective as defined in Cycle 220
Q216=+90 ;CENTER IN 1ST AXIS	
Q217=+25 ;CENTER IN 2ND AXIS	
Q244=70 ;PITCH CIRCLE DIAMETR	
Q245=+90 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+30 ;STEPPING ANGLE	
Q241=5 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=0 ;TYPE OF TRAVERSE	
8 L Z+250 R0 FMAX M2	Retract the tool, end program
9 END PGM HOLEPAT MM	

8

Fixed Cycles: Contour Pocket

8.1 SL Cycles

Fundamentals

SL cycles enable you to form complex contours by combining up to twelve subcontours (pockets or islands). You define the individual subcontours in subprograms. The control calculates the entire contour from the subcontours (subprogram numbers) that you enter in Cycle 14 CONTOUR.

> The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
>
> SL cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, you should always run a graphical program test before machining! This is a simple way of finding out whether the program calculated by the control will provide the desired results.
>
> If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The control recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR
- The control recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL
- The subprograms must not contain spindle axis coordinates.
- Always program both axes in the first NC block of the subprogram
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms

Program structure: Machining with SL cycles

```
0 BEGIN PGM SL2 MM
...
12 CYCL DEF 14 CONTOUR ...
13 CYCL DEF 20 CONTOUR DATA ...
...
16 CYCL DEF 21 PILOT DRILLING ...
17 CYCL CALL
...
18 CYCL DEF 22 ROUGH-OUT ...
19 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING ...
23 CYCL CALL
...
26 CYCL DEF 24 SIDE FINISHING ...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
```

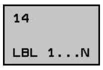
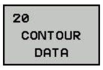
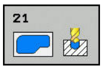
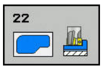
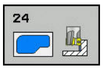
```
60 LBL 0
...
99 END PGM  SL2 MM
```

Characteristics of the fixed cycles

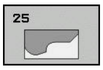
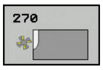
- The control automatically positions the tool to the set-up clearance before each cycle. You must move the tool to a safe position before the cycle call
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them
- The radius of inside 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)	231
	20 CONTOUR DATA (compulsory)	236
	21 PILOT DRILLING (optional)	238
	22 ROUGH-OUT (compulsory)	240
	23 FLOOR FINISHING (optional)	244
	24 SIDE FINISHING (optional)	246

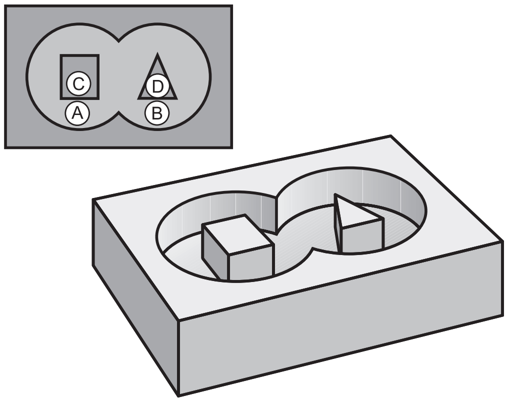
Enhanced cycles:

Soft key	Cycle	Page
	25 CONTOUR TRAIN	249
	270 CONTOUR TRAIN DATA	258

8.2 CONTOUR (Cycle 14, ISO: G37)

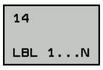
Please note while programming:

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



> Cycle 14 is DEF-active which means that it becomes effective as soon as it is defined in the NC program.
>
> You can list up to 12 subprograms (subcontours) in Cycle 14.

Cycle parameters

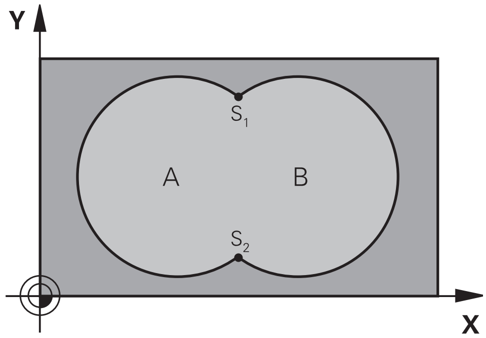


- **Label numbers for the contour**: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm each input with the ENT key. Confirm each input with the **END** key. Input of up to 12 subprogram numbers: 1 to 65 535

8.3 Superimposed contours

Fundamentals

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.



Example

12 CYCL DEF 14.0 CONTOUR
13 CYCL DEF 14.1 CONTOUR LABEL1/2/3/4

Subprograms: overlapping pockets

> The following examples show contour subprograms that are called by Cycle 14 CONTOUR in a main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2. They need not be programmed.

The pockets are programmed as full circles.

Subprogram 1: Pocket A

51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0

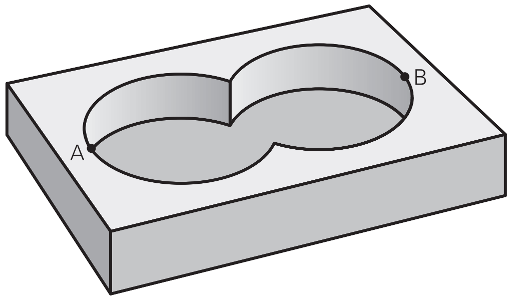
Subprogram 2: Pocket B

56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0

Area of inclusion

Both surfaces A and B are to be machined, including the overlapping area:

- The surfaces A and B must be pockets
- The first pocket (in Cycle 14) must start outside the second pocket



Surface A:

51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0

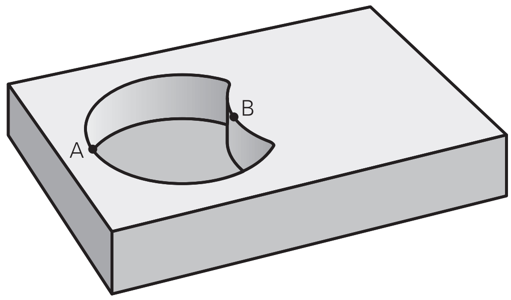
Surface B:

56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0

Area of exclusion

Surface A is to be machined without the portion overlapped by B:

- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.



Surface A:

51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0

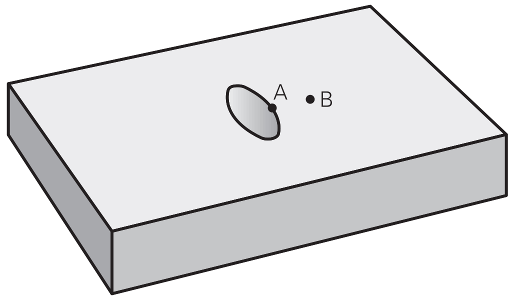
Surface B:

56 LBL 2
57 L X+40 Y+50 RL
58 CC X+65 Y+50
59 C X+40 Y+50 DR-
60 LBL 0

Area of intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- A and B must be pockets
- A must start inside of B



Surface A:

```
51 LBL 1
52 L  X+60 Y+50  RR
53 CC  X+35 Y+50
54 C  X+60 Y+50  DR-
55 LBL 0
```

Surface B:

```
56 LBL 2
57 L  X+90 Y+50  RR
58 CC  X+65 Y+50
59 C  X+90 Y+50  DR-
60 LBL 0
```

8.4 CONTOUR DATA (Cycle 20, ISO: G120)

Please note while programming:

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

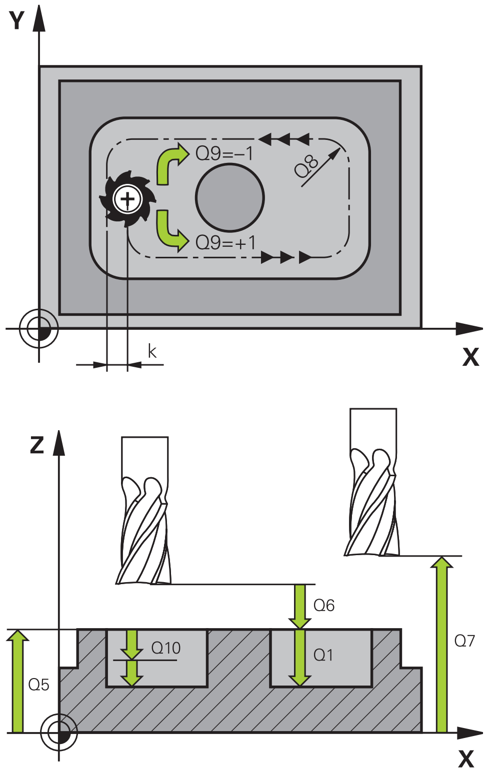
> Cycle 20 is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
>
> The machining data entered in Cycle 20 are valid for Cycles 21 to 24.
>
> The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH = 0, the control 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

20 CONTOUR DATA

- **Q1 Milling depth?** (incremental): Distance between workpiece surface and bottom of pocket. Input range: -99999.9999 to 99999.9999
- **Q2 Path overlap factor?**: Q2 x tool radius = stepover factor k. Input range: -0.0001 to 1.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: -99999.9999 to 99999.9999
- **Q4 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: -99999.9999 to 99999.9999
- **Q5 Workpiece surface coordinate?** (absolute): Absolute coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q6 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Input range: 0 to 99999.9999
- **Q7 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: -99999.9999 to 99999.9999
- **Q8 Inside corner radius?**: Inside "corner" rounding radius; entered value references the path of the tool center and is used to calculate smoother traverse motions between the contour elements. **Q8 is not a radius that is inserted as a separate contour element between programmed elements!** Input range: 0 to 99999.9999
- **Q9 Direction of rotation? cw = -1**: Machining direction for pockets
  - Q9 = –1 up-cut milling for pocket and island
  - Q9 = +1 climb milling for pocket and island

You can check the machining parameters during a program interruption and overwrite them, if required.



Example

57 CYCL DEF 20 CONTOUR DATA	
Q1=-20	;MILLING DEPTH
Q2=1	;TOOL PATH OVERLAP
Q3=+0.2	;ALLOWANCE FOR SIDE
Q4=+0.1	;ALLOWANCE FOR FLOOR
Q5=+30	;SURFACE COORDINATE
Q6=2	;SET-UP CLEARANCE
Q7=+80	;CLEARANCE HEIGHT
Q8=0.5	;ROUNDING RADIUS
Q9=+1	;ROTATIONAL DIRECTION

8.5 PILOT DRILLING (Cycle 21, ISO: G121)

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:

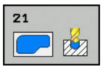
When calculating the infeed points, the control does not account for the delta value **DR** programmed in a **TOOL CALL** block.

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

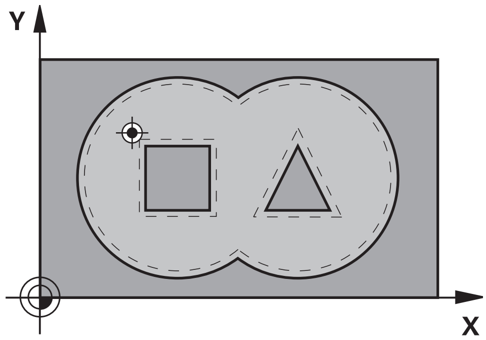
If Q13=0, the TNC uses the data of the tool that is currently in the spindle.

After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have set the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter to **ToolAxClearanceHeight**.

Cycle parameters



- **Q10 Plunging depth?** (incremental): Dimension by which the tool drills in each infeed (minus sign for negative working direction). Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q13 Rough-out tool number/name?** or QS13: Number or name of rough-out tool. It is possible to take over the tool via soft key directly from the tool table.

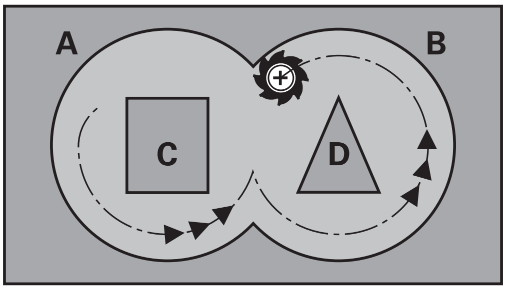


Example

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

8.6 ROUGHING (Cycle 22, ISO: G122)

Cycle run



Use Cycle 22 ROUGHING to define the technology data for roughing.

Before calling Cycle 22 you need to program further cycles:

- Cycle 14 CONTOUR or SEL CONTOUR
- Cycle 20 CONTOUR DATA
- Cycle 21 PILOT DRILLING, if necessary

Cycle run

1. The control positions the tool above the cutter infeed point, taking the allowance for side into account
2. In the first plunging depth, the tool mills the contour from inside outward at the milling feed rate.
3. First the island contours (C and D in the figure at right) are rough-milled until the pocket contour (A, B) is approached.
4. In the next step, the control moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This depends on the **ConfigDatum**, **CfgGeoCycle** (no. 201000), **posAfterContPocket** (no. 201007) parameter.

Please note while programming:

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

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

- If Q19=0 is defined, the tool will always plunge perpendicularly, even if a plunge angle (**ANGLE**) was defined for the active tool
- If you define **ANGLE**=90°, the control will plunge perpendicularly. The reciprocation feed rate Q19 is used as plunging feed rate
- If the reciprocation feed rate Q19 is defined in Cycle 22 and **ANGLE** is between 0.1 and 89.999 in the tool table, the control plunges helically using the defined **ANGLE**
- If the reciprocation feed is defined in Cycle 22 and no **ANGLE** can be found in the tool table, the control displays an error message
- If geometrical conditions do not allow helical plunging (slot geometry), the control tries a reciprocating plunge. The reciprocation length is calculated from **LCUTS** and **ANGLE** (reciprocation length = **LCUTS** / tan **ANGLE**)

If you clear out an acute inside corner and use an overlap factor greater than 1, some material might be left over. Check especially the innermost path in the test run graphic and, if necessary, change the overlap factor slightly. This allows another distribution of cuts, which often provides the desired results.

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

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

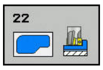
NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- ▶ After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- ▶ Make sure to program an absolute position after the cycle, no incremental traverse

Cycle parameters



- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion that is to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered
- **Q19 Feed rate for reciprocation?**: Traversing speed of the tool in mm/min during reciprocating plunge-cut. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the control retracts the tool at the feed rate specified in Q12. Input range: 0 to 99999,9999; alternatively **FMAX, FAUTO**

Example

59 CYCL DEF 22 ROUGH-OUT	
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=750	;FEED RATE F. ROUGHNG
Q18=1	;COARSE ROUGHING TOOL
Q19=150	;FEED RATE FOR RECIP.
Q208=9999	;RETRACTION FEED RATE
Q401=80	;FEED RATE FACTOR
Q404=0	;FINE ROUGH STRATEGY

- **Q401 Feed rate factor in %?**: Percentage factor by which the control reduces the machining feed rate (Q12) as soon as the tool moves with its entire circumference within the material during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap (Q2) specified in Cycle 20. The control then reduces the feed rate as per your definition at transitions and narrow places, reducing the total machining time. Input range: 0.0001 to 100.0000
- **Q404 Fine roughing strategy (0/1)?**: Specify the fine roughing behavior of the control if the radius of the fine roughing tool is greater than or equal to half the radius of the coarse roughing tool:
  Q404=0:
  The control moves the tool between the areas to be fine-roughed at the current depth along the contour
  Q404=1:
  The control retracts the tool to the set-up clearance between the areas to be fine-roughed and then moves to the starting point for the next area to be roughed-out

8.7 FLOOR FINISHING (Cycle 23, ISO: G123)

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:

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

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

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

NOTICE

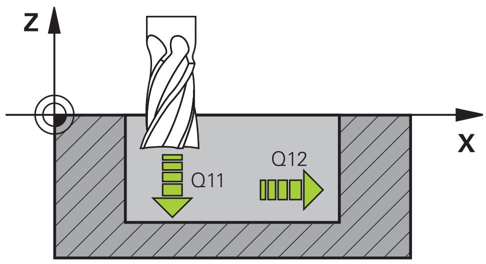
Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

Cycle parameters

- **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q208 Feed rate for retraction?**: Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter Q208 = 0, the control retracts the tool at the feed rate specified in Q12. Input range: 0 to 99999,9999; alternatively **FMAX, FAUTO**



Example

60 CYCL DEF 23 FLOOR FINISHING	
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q208=9999	;RETRACTION FEED RATE

8.8 SIDE FINISHING (Cycle 24, ISO: G124)

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:

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

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

The allowance for side Q14 is left over after finishing. Therefore, it must be smaller than the allowance in Cycle 20.

This calculation also holds if you run Cycle 24 without having roughed out with Cycle 22; in this case, enter "0" for the radius of the rough mill.

Cycle 24 can also be used for contour milling. In that case, you must do the following:

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

The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket and the allowance programmed in Cycle 20.

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

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

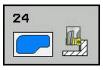
NOTICE

Danger of collision!

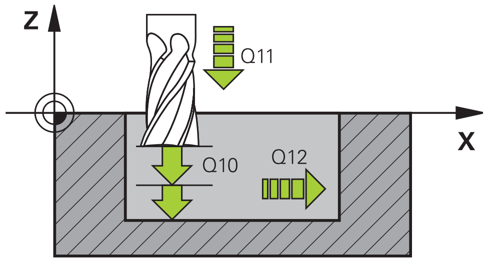
If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

Cycle parameters



- **Q9 Direction of rotation? cw = -1**: Machining direction:
  +1: Counterclockwise rotation
  -1: Clockwise rotation
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in mm/min during plunging. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q14 Finishing allowance for side?** (incremental): The allowance for side Q14 is left over after finishing. (This allowance must be smaller than the allowance in Cycle 20.) Input range: -99999.9999 to 99999.9999
- **Q438 Number/name of the rough-out tool Q438** or **QS438**: Number or name of the tool with which the control roughed out the contour pocket. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. Input range if a number is entered: -1 to +32767.9
  Q438=-1: The control assumes that the tool last used was the rough-out tool (default behavior)
  Q438=0: If there was no coarse-roughing, enter 0. A roughing tool with the radius 0 is assumed

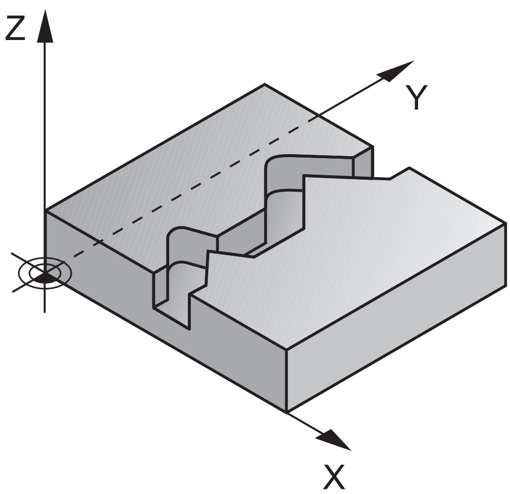


Example

61 CYCL DEF 24 SIDE FINISHING	
Q9=+1	;ROTATIONAL DIRECTION
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q14=+0	;ALLOWANCE FOR SIDE
Q438=-1	;NUMBER/NAME OF ROUGH-OUT TOOL?

8.9 CONTOUR TRAIN (Cycle 25, ISO: G125)

Cycle run



In conjunction with Cycle 14 CONTOUR, this cycle facilitates the machining of open and closed contours.

Cycle 25 CONTOUR TRAIN offers considerable advantages over machining a contour using positioning blocks:

- The control monitors the operation to prevent undercuts and contour damage. It is recommended that you run a graphic simulation of the contour before execution
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked
- The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when the contours are mirrored.
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.

Please note while programming!

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

The control takes only the first label of Cycle 14 CONTOUR into account.

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

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

Cycle 20 **CONTOUR DATA** is not required.

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

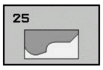
NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

Cycle parameters



- **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: -99999.9999 to 99999.9999
- **Q5 Workpiece surface coordinate?** (absolute): Absolute coordinate of the workpiece surface. Input range: -99999.9999 to 99999.9999
- **Q7 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: -99999.9999 to 99999.9999
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q15 Climb or up-cut? up-cut = -1**:
  Climb milling: Input value = +1
  Up-cut milling: Input value = –1
  Climb milling and up-cut milling alternately in several infeeds: Input value = 0

Example

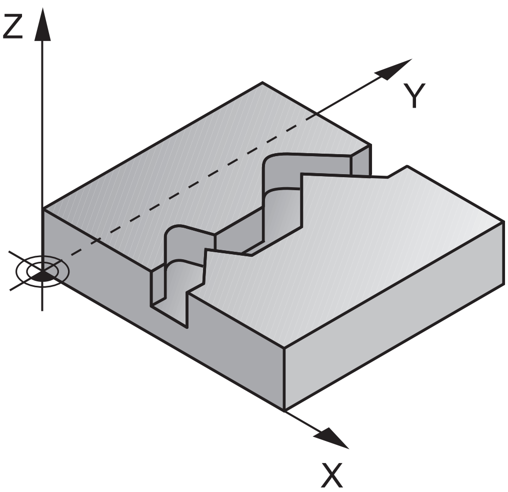
62 CYCL DEF 25 CONTOUR TRAIN	
Q1=-20	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q5=+0	;SURFACE COORDINATE
Q7=+50	;CLEARANCE HEIGHT
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q15=-1	;CLIMB OR UP-CUT
Q18=0	;COARSE ROUGHING TOOL
Q446=+0.01	;RESIDUAL MATERIAL
Q447=+10	;CONNECTION DISTANCE
Q448=+2	;PATH EXTENSION

- **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion that is to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered
- **Q446 Accepted residual material?** Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input range 0.001 to 9.999
- **Q447 Maximum connection distance?** Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input range: 0 to 999.9999
- **Q448 Path extension?** Length by which the tool path is extended at the start and end of the contour. The control always extends the tool path in parallel to the contour. Input range 0 to 99.999

8.10 THREE-D CONT. TRAIN (Cycle 276, ISO: G276)

Cycle run

In conjunction with Cycle 14 CONTOUR and Cycle 270 CONTOUR TRAIN DATA, this cycle facilitates the machining of open and closed contours. You can also work with automatic residual material detection. This way you can subsequently complete e.g. inside corners with a smaller tool.



In contrast to Cycle 25 CONTOUR TRAIN, Cycle 276 THREE-D CONT. TRAIN also processes tool axis coordinates defined in the contour subprogram. This cycle can thus machine three-dimensional contours.

We recommend that you program Cycle 270 CONTOUR TRAIN DATA before Cycle 276 THREE-D CONT. TRAIN.

Machining a contour without infeed: Milling depth Q1=0

1. The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle 270 CONTOUR TRAIN DATA, e.g. in Start t.. The control then moves the tool to the first plunging depth
2. According to the previously defined Cycle 270 CONTOUR TRAIN DATA, the tool approaches the contour and then machines it completely to the end
3. At the end of the contour, the tool will be retracted as defined in Cycle 270 CONTOUR TRAIN DATA
4. Finally, the control retracts the tool to the clearance height.

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

1. The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle 270 CONTOUR TRAIN DATA, e.g. in Start t.. The control then moves the tool to the first plunging depth
2. According to the previously defined Cycle 270 CONTOUR TRAIN DATA, the tool approaches the contour and then machines it completely to the end
3. If you selected machining with Climb milling and up-cut milling (Q15=0), the control will perform a reciprocation movement. The infeed movement (plunging) will be performed at the end and at the starting point of the contour. If Q15 is not equal to 0, the tool is moved to clearance height and returned to the starting point of machining. From there, the control moves the tool to the next plunging depth
4. The departure will be performed as defined in Cycle 270 CONTOUR TRAIN DATA
5. This process is repeated until the programmed depth is reached
6. Finally, the control retracts the tool to the clearance height.

Please note while programming:

The first NC block in the contour subprogram must contain values in all of the three axes X, Y and Z.

If you program **APPR** and **DEP** blocks for contour approach and departure, the control monitors whether the execution of any of these blocks would damage the contour

The algebraic sign for the depth parameter determines the working direction. If you program DEPTH=0, the cycle will use the tool axis coordinates that have been specified in the contour subprogram.

If using Cycle 25 CONTOUR TRAIN, you can define only one subprogram in the CONTOUR cycle.

We recommend that you use Cycle 270 CONTOUR TRAIN DATA in conjunction with Cycle 276. Cycle 20 CONTOUR DATA, however, is not required.

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

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

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

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

NOTICE

Danger of collision!

A collision may occur if you position the tool behind an obstacle before the cycle is called.

- Before the cycle call, position the tool in such a way that the tool can approach the starting point of the contour without collision
- If the position of the tool is below the clearance height when the cycle is called, the control will issue an error message

Cycle parameters

- **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range: -99999.9999 to 99999.9999
- **Q7 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: -99999.9999 to 99999.9999
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q15 Climb or up-cut? up-cut = -1**:
  Climb milling: Input value = +1
  Up-cut milling: Input value = –1
  Climb milling and up-cut milling alternately in several infeeds: Input value = 0
- **Q18 Coarse roughing tool?** or **QS18**: Number or name of the tool with which the control has already coarse-roughed the contour. It is possible to take over the coarse roughing tool via soft key directly from the tool table. In addition, the tool name can be entered via the **tool name** soft key. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion that is to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE** there. Input range: 0 to 99999 if a number is entered; maximum 16 characters if a name is entered

Example

62 CYCL DEF 276 THREE-D CONT. TRAIN	
Q1=-20	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q7=+50	;CLEARANCE HEIGHT
Q10=-5	;PLUNGING DEPTH
Q11=150	;FEED RATE FOR PLNGNG
Q12=500	;FEED RATE F. ROUGHNG
Q15=+1	;CLIMB OR UP-CUT
Q18=0	;COARSE ROUGHING TOOL
Q446=+0.01	;RESIDUAL MATERIAL
Q447=+10	;CONNECTION DISTANCE
Q448=+2	;PATH EXTENSION

- **Q446 Accepted residual material?** Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input range 0.001 to 9.999
- **Q447 Maximum connection distance?** Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input range: 0 to 999.9999
- **Q448 Path extension?** Length by which the tool path is extended at the start and end of the contour. The control always extends the tool path in parallel to the contour. Input range 0 to 99.999

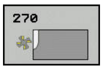
8.11 CONTOUR TRAIN DATA (Cycle 270, ISO: G270)

Please note while programming:

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

> Cycle 270 is DEF-active, which means that it becomes effective as soon as it is defined in the NC program.
>
> If Cycle 270 is used, do not define any radius compensation in the contour subprogram.
>
> Define Cycle 270 before Cycle 25.

Cycle parameters



- **Q390 Type of approach/departure?**: Definition of the type of approach or departure:
  Q390=1:
  Approach the contour tangentially on a circular arc
  Q390=2:
  Approach the contour tangentially on a straight line
  Q390=3:
  Approach the contour at a right angle
- **Q391 Radius comp. (0=R0/1=RL/2=RR)?**: Definition of the radius compensation:
  Q391=0:
  Machine the defined contour without radius compensation
  Q391=1:
  Machine the defined contour with compensation to the left
  Q391=2:
  Machine the defined contour with compensation to the right
- **Q392 App. radius/dep. radius?**: Only in effect if a tangential approach on a circular path was selected (Q390 = 1). Radius of the approach/departure arc. Input range: 0 to 99999.9999
- **Q393 Center angle?**: Only in effect if a tangential approach on a circular path was selected (Q390 = 1). Angular length of the approach arc. Input range: 0 to 99999.9999
- **Q394 Distance from aux. point?**: Only in effect if a tangential approach on a straight line or a right-angle approach is selected (Q390=2 or Q390=3). Distance to the auxiliary point from which the tool is to approach the contour. Input range: 0 to 99999.9999

Example

62 CYCL DEF 270 CONTOUR TRAIN DATA	
Q390=1	;TYPE OF APPROACH
Q391=1	;RADIUS COMPENSATION
Q392=3	;RADIUS
Q393=+45	;CENTER ANGLE
Q394=+2	;DISTANCE

8.12 TROCHOIDAL SLOT (Cycle 275, ISO: G275)

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. Enormous amounts of time can also be saved by combining this milling method with the integrated adaptive feed control **AFC** (software option).(**Further information**: Conversational Programming User's Manual)

Depending on the cycle parameters you select, the following machining alternatives are available:

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

Roughing with closed slots

In case of a closed slot, the contour description must always start with an straight line block (**L** block).

1 Following the positioning logic, the tool moves to the starting point of the contour description and moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached.

Finishing with closed slots

5 If a finishing allowance has been defined, the control finishes the slot walls, in multiple infeeds, if so specified. Starting from the defined starting point, the control approaches the slot wall tangentially. Climb or up-cut milling is taken into consideration.

Program structure: Machining with SL cycles

0 BEGIN PGM CYC275 MM
...
12 CYCL DEF 14.0 CONTOUR
13 CYCL DEF 14.1 CONTOUR LABEL 10
14 CYCL DEF 275 TROCHOIDAL SLOT...
15 CYCL CALL M3
...
50 L Z+250 R0 FMAX M2
51 LBL 10
...
55 LBL 0
...
99 END PGM CYC275 MM

Roughing with open slots

The contour description of an open slot must always start with an approach block (**APPR**).

1 Following the positioning logic, the tool moves to the starting point of the machining operation as defined by the parameters in the **APPR** block and positions there perpendicular to the first plunging depth.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached.

Finishing with open slots

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

Please note while programming:

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

When using Cycle 275 TROCHOIDAL SLOT, you can define only one contour subprogram in Cycle 14 CONTOUR.

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

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

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

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

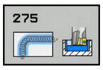
NOTICE

Danger of collision!

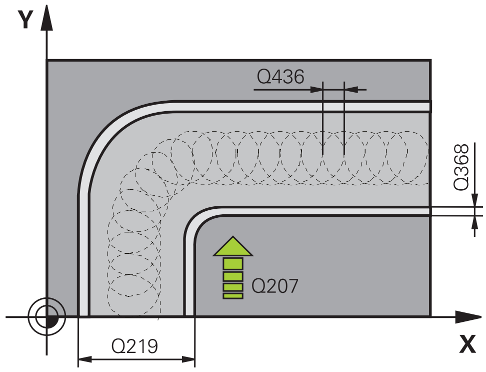
If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

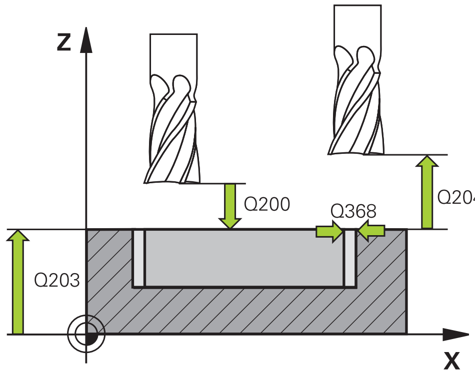
- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle, no incremental traverse

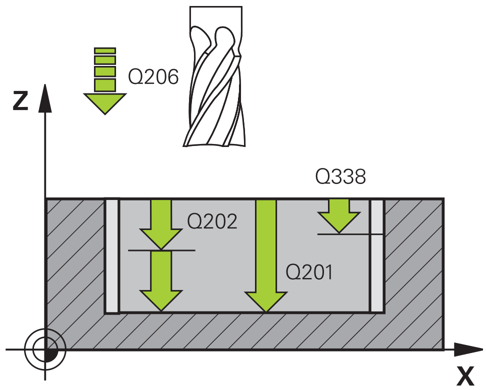
Cycle parameters



- **Q215 Machining operation (0/1/2)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
  Side finishing and floor finishing are only carried out if the required finishing allowance (Q368, Q369) has been programmed
- **Q219 Width of slot?** (value parallel to the minor axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the control will carry out the roughing process only (oblong hole milling). Maximum slot width for roughing: Twice the tool diameter. Input range: 0 to 99999.9999
- **Q368 Finishing allowance for side?** (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Q436 Feed per revolution?** (absolute): Value by which the control moves the tool in the machining direction per revolution. Input range: 0 to 99999.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q351 Direction? Climb=+1, Up-cut=-1**: Type of milling operation with M3:
  +1 = Climb
  -1 = Up-cut
  PREDEF: The control uses the value from the GLOBAL DEF block (if you enter 0, climb milling is performed)
- **Q201 Depth?** (incremental): Distance between workpiece surface and bottom of slot. Input range: -99999.9999 to 99999.9999







- **Q202 Plunging depth?** (incremental): Infeed per cut; enter a value greater than 0. Input range: 0 to 99999.9999
- **Q206 Feed rate for plunging?**: Traversing speed of the tool in mm/min when plunging to depth. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**
- **Q338 Infeed for finishing?** (incremental): Infeed in the spindle axis per finishing cut. Q338=0: Finishing in one infeed. Input range: 0 to 99999.9999
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min during side and floor finishing. Input range: 0 to 99999,999; alternatively **FAUTO**, **FU**, **FZ**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Q366 Plunging strategy (0/1/2)?**: Type of plunging strategy:
  0 = vertical plunging. The control plunges the tool perpendicularly, regardless of the plunging angle ANGLE defined in the tool table
  1 = No function
  2 = reciprocating plunge. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message
  Alternatively: **PREDEF**

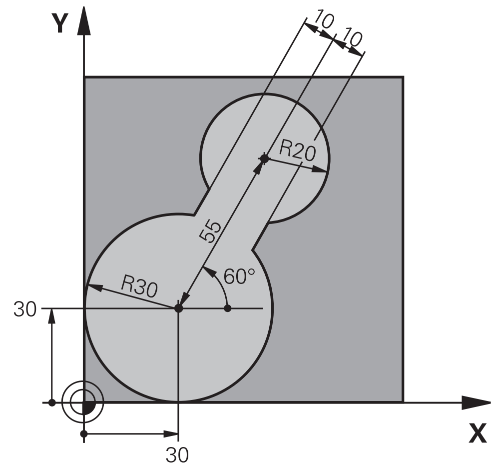
Example

```
8 CYCL DEF 275 TROCHOIDAL SLOT
    Q215=0      ;MACHINING OPERATION
    Q219=12     ;SLOT WIDTH
    Q368=0.2    ;ALLOWANCE FOR SIDE
    Q436=2      ;INFEED PER REV.
    Q207=500    ;FEED RATE FOR MILLNG
    Q351=+1     ;CLIMB OR UP-CUT
    Q201=-20    ;DEPTH
    Q202=5      ;PLUNGING DEPTH
    Q206=150    ;FEED RATE FOR PLNGNG
    Q338=5      ;INFEED FOR FINISHING
    Q385=500    ;FINISHING FEED RATE
    Q200=2      ;SET-UP CLEARANCE
    Q203=+0     ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
    Q366=2      ;PLUNGE
    Q369=0      ;ALLOWANCE FOR FLOOR
    Q439=0      ;FEED RATE REFERENCE
9 CYCL CALL FMAX M3
```

- **Q369 Finishing allowance for floor?** (incremental): Finishing allowance for the floor. Input range: 0 to 99999.9999
- **Q439 Feed rate reference (0-3)?**: Specify what the programmed feed rate refers to:
  0: Feed rate with respect to the tool center point path
  1: Feed rate with respect to the tool edge, but only during side finishing, otherwise with respect to the tool center path
  2: Feed rate refers to the tool cutting edge during side finishing **and** floor finishing; otherwise, it refers to the tool center path
  3: Feed rate always refers to the cutting edge

8.13 Programming Examples

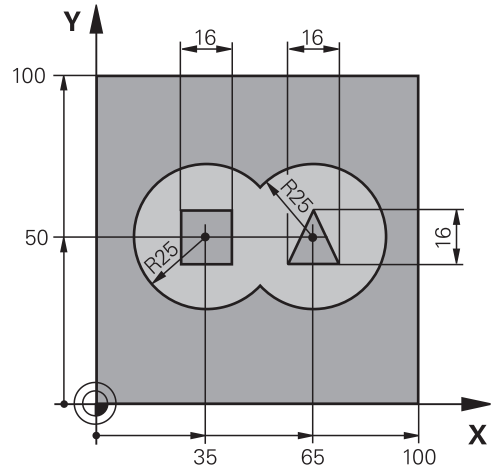
Example: Roughing-out and fine-roughing a pocket



0 BEGIN PGM C20 MM	
1 BLK FORM 0.1 Z X-10 Y-10 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	Workpiece blank definition
3 TOOL CALL 1 Z S2500	Tool call: coarse roughing tool, diameter 30
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 22 ROUGH-OUT	Cycle definition: coarse roughing
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE F. ROUGHNG	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q208=30000 ;RETRACTION FEED RATE	
9 CYCL CALL M3	Cycle call: coarse roughing
10 L Z+250 R0 FMAX M6	Retract the tool

11 TOOL CALL 2 Z S3000	Tool call: fine roughing tool, diameter 15
12 CYCL DEF 22 ROUGH-OUT	Cycle definition: fine roughing
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE F. ROUGHNG	
Q18=1 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q208=30000 ;RETRACTION FEED RATE	
13 CYCL CALL M3	Cycle call: fine roughing
14 L Z+250 R0 FMAX M2	Retract the tool, end program
15 LBL 1	Contour subprogram
16 L X+0 Y+30 RR	
17 FC DR- R30 CCX+30 CCY+30	
18 FL AN+60 PDX+30 PDY+30 D10	
19 FSELECT 3	
20 FPOL X+30 Y+30	
21 FC DR- R20 CCPR+55 CCPA+60	
22 FSELECT 2	
23 FL AN-120 PDX+30 PDY+30 D10	
24 FSELECT 3	
25 FC X+0 DR- R30 CCX+30 CCY+30	
26 FSELECT 2	
27 LBL 0	
28 END PGM C20 MM	

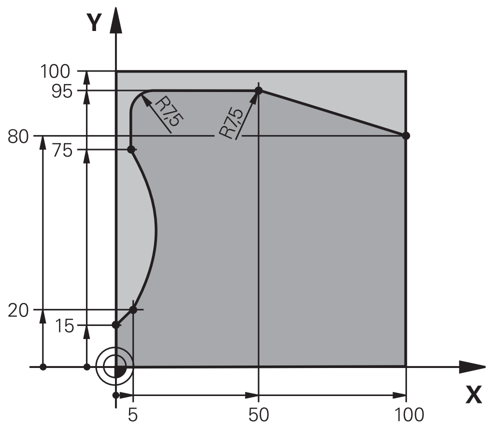
Example: Pilot drilling, roughing-out and finishing overlapping contours



0 BEGIN PGM C21 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: drill, diameter 12
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1/2/3/4	
7 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 21 PILOT DRILLING	Cycle definition: pilot drilling
Q10=5 ;PLUNGING DEPTH	
Q11=250 ;FEED RATE FOR PLNGNG	
Q13=2 ;ROUGH-OUT TOOL	
9 CYCL CALL M3	Cycle call: pilot drilling
10 L +250 R0 FMAX M6	Retract the tool
11 TOOL CALL 2 Z S3000	Tool call: roughing/finishing, diameter 12
12 CYCL DEF 22 ROUGH-OUT	Cycle definition: rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	

Q12=350 ;FEED RATE F. ROUGHNG	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q208=30000 ;RETRACTION FEED RATE	
13 CYCL CALL M3	Cycle call: rough-out
14 CYCL DEF 23 FLOOR FINISHING	Cycle definition: floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE F. ROUGHNG	
Q208=30000 ;RETRACTION FEED RATE	
15 CYCL CALL	Cycle call: floor finishing
16 CYCL DEF 24 SIDE FINISHING	Cycle definition: side finishing
Q9=+1 ;ROTATIONAL DIRECTION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE F. ROUGHNG	
Q14=+0 ;ALLOWANCE FOR SIDE	
17 CYCL CALL	Cycle call: side finishing
18 L Z+250 R0 FMAX M2	Retract the tool, end program
19 LBL 1	Contour subprogram 1: left pocket
20 CC X+35 Y+50	
21 L X+10 Y+50 RR	
22 C X+10 DR-	
23 LBL 0	
24 LBL 2	Contour subprogram 2: right pocket
25 CC X+65 Y+50	
26 L X+90 Y+50 RR	
27 C X+90 DR-	
28 LBL 0	
29 LBL 3	Contour subprogram 3: left square island
30 L X+27 Y+50 RL	
31 L Y+58	
32 L X+43	
33 L Y+42	
34 L X+27	
35 LBL 0	
36 LBL 4	Contour subprogram 4: right triangular island
37 L X+65 Y+42 RL	
38 L X+57	
39 L X+65 Y+58	
40 L X+73 Y+42	
41 LBL 0	
42 END PGM C21 MM	

Example: Contour train



0 BEGIN PGM C25 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2000	Tool call: diameter 20
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 25 CONTOUR TRAIN	Define machining parameters
Q1=-20 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q5=+0 ;SURFACE COORDINATE	
Q7=+250 ;CLEARANCE HEIGHT	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE F. ROUGHNG	
Q15=+1 ;CLIMB OR UP-CUT	
Q466= 0.01 ;RESIDUAL MATERIAL	
Q447=+10 ;CONNECTION DISTANCE	
Q448=+2 ;PATH EXTENSION	
8 CYCL CALL M3	Cycle call
9 L Z+250 R0 FMAX M2	Retract the tool, end program
10 LBL 1	Contour subprogram
11 L X+0 Y+15 RL	
12 L X+5 Y+20	
13 CT X+5 Y+75	
14 L Y+95	
15 RND R7.5	
16 L X+50	
17 RND R7.5	

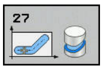
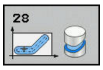
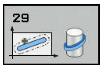
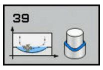
18 L X+100 Y+80	
19 LBL 0	
20 END PGM C25 MM	

9

Fixed Cycles: Cylindrical Surface

9.1 Fundamentals

Overview of cylindrical surface cycles

Soft key	Cycle	Page
	27 CYLINDER SURFACE	273
	28 CYLINDER SURFACE Slot milling	276
	29 CYLINDER SURFACE Ridge milling	280
	39 CYLINDER SURFACE Contour	283

9.2 CYLINDER SURFACE (Cycle 27, ISO: G127, software option 1)

Cycle run

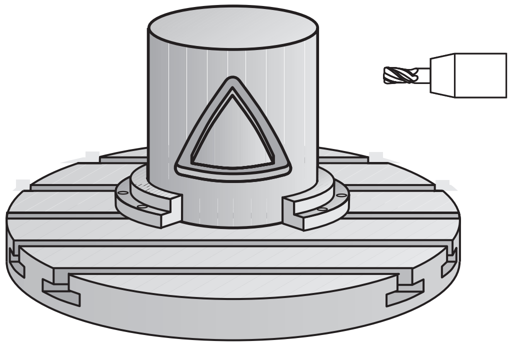
This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle 28 if you want to mill guideways on the cylinder.

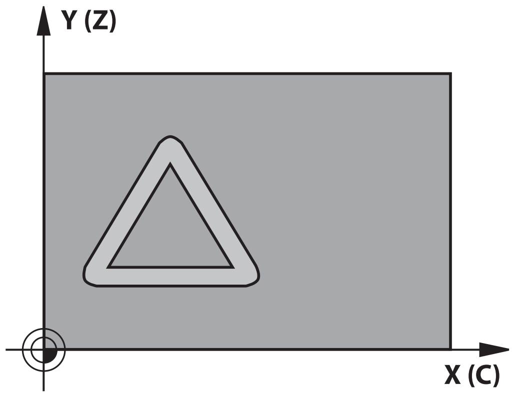
The contour is described in a subprogram identified in Cycle 14 CONTOUR.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

The dimensions in the rotary axis (x coordinates) can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition using Q17.

1 The control positions the tool above the cutter infeed point, taking the allowance for side into account
2 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12.
3 At the end of the contour, the control returns the tool to the set-up clearance and returns to the infeed point
4 Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.
5 Subsequently, the tool retracts in the tool axis to the clearance height.





Please note while programming:

Refer to your machine manual.

The machine manufacturer must prepare the machine and the control for cylinder surface interpolation.

In the first NC block of the contour program, always program both cylinder surface coordinates.

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

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

This cycle requires a center-cut end mill (ISO 1641).

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

The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.

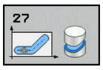
This cycle can also be used in a tilted working plane.

The set-up clearance must be greater than the tool radius.

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

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

Cycle parameters



- **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the plane of the developed cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range: -99999.9999 to 99999.9999
- **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999
- **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches

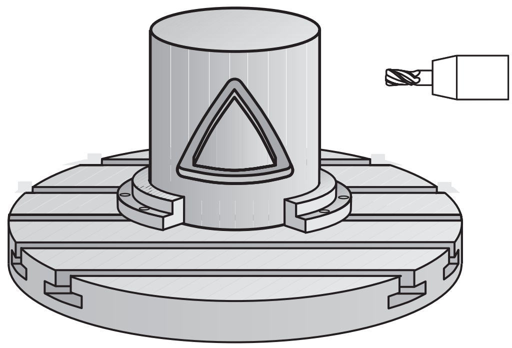
Example

63 CYCL DEF 27 CYLINDER SURFACE	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION

9.3 CYLINDER SURFACE Slot milling (Cycle 28, ISO: G128, software option 1)

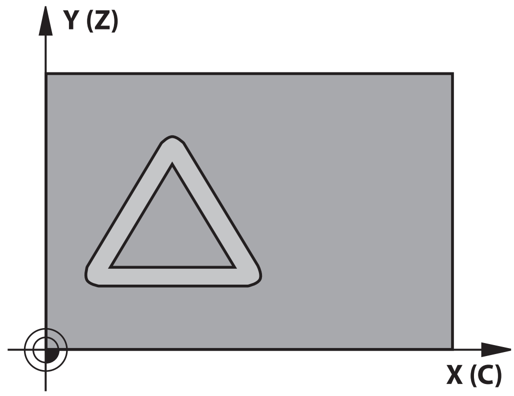
Cycle run

With this cycle you can program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle 27, with this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the slot are nearly parallel. You can machine exactly parallel walls by using a tool that is exactly as wide as the slot.



The smaller the tool is with respect to the slot width, the larger the distortion in circular arcs and oblique line segments. To minimize this process-related distortion, you can define the parameter Q21. This parameter specifies the tolerance with which the control machines a slot as similar as possible to a slot machined with a tool of the same width as the slot.

Program the center path of the contour together with the tool radius compensation. With the radius compensation you specify whether the control cuts the slot with climb milling or up-cut milling.



1 The control positions the tool above the infeed point.
2 The control moves the tool vertically to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate Q12. The approaching behavior depends on the **ConfigDatum CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004) parameter
3 At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate Q12 while respecting the finishing allowance for the side
4 At the end of the contour, the control moves the tool to the opposite slot wall and returns to the infeed point.
5 Steps 2 to 3 are repeated until the programmed milling depth Q1 is reached.
6 If you have defined the tolerance in Q21, the control then re-machines the slot walls to be as parallel as possible.
7 Finally, the tool retracts in the tool axis to the clearance height.

Please note while programming:

This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

Define the approaching behavior in **ConfigDatum**, **CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004)

- CircleTangential:
  Tangential approach and departure
- LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

In the first NC block of the contour program, always program both cylinder surface coordinates.

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

This cycle requires a center-cut end mill (ISO 1641).

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

The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

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

The set-up clearance must be greater than the tool radius.

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

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

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** parameter (no. 201002), to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.
- The function needs to be adapted by your machine manufacturer.

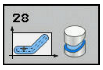
NOTICE

Danger of collision!

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

- Control the traversing movements of the machine
- In the simulation, control the end position of the tool after the cycle
- After the cycle, program the absolute (not incremental) coordinates

Cycle parameters



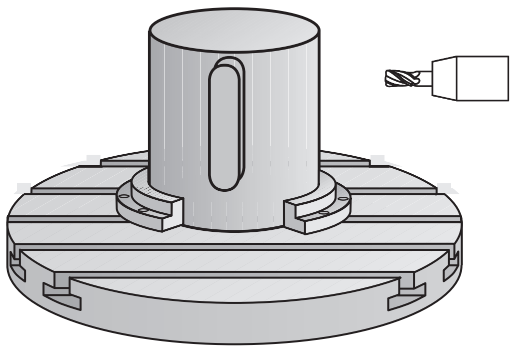
- **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. Input range: -99999.9999 to 99999.9999
- **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999
- **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches
- **Q20 Slot width?**: Width of the slot to be machined. Input range: -99999.9999 to 99999.9999
- **Q21 Tolerance?**: If you use a tool smaller than the programmed slot width Q20, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance Q21, the control adds a subsequent milling operation to ensure that the slot dimensions are as close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With Q21, you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the re-machining takes. Input range for tolerance: 0.0001 to 9.9999
  Recommendation: Use a tolerance of 0.02 mm.
  Function inactive: Enter 0 (default setting).

Example

```
63 CYCL DEF 28  CYLINDER SURFACE
    Q1=-8       ;MILLING DEPTH
    Q3=+0       ;ALLOWANCE FOR SIDE
    Q6=+0       ;SET-UP CLEARANCE
    Q10=+3      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLNGNG
    Q12=350     ;FEED RATE F. ROUGHNG
    Q16=25      ;RADIUS
    Q17=0       ;TYPE OF DIMENSION
    Q20=12      ;SLOT WIDTH
    Q21=0       ;TOLERANCE
```

9.4 CYLINDER SURFACE Ridge milling (Cycle 29, ISO: G129, software option 1)

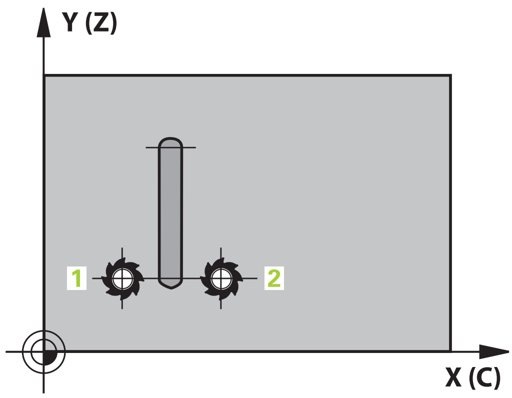
Cycle run



This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. With this cycle, the control adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the center path of the ridge together with the tool radius compensation. With the radius compensation you specify whether the control cuts the ridge with climb milling or up-cut milling.

At the ends of the ridge, the control will always add a semi-circle whose radius corresponds to half the ridge width.

1 The control positions the tool above the starting point of machining. The control calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins to the left (**1**, RL = climb milling) or to the right of the ridge (**2**, RR = up-cut milling).
2 After the control has positioned the tool to the first plunging depth, the tool moves on a circular arc at the milling feed rate Q12 tangentially to the ridge wall. A finishing allowance programmed for the side is taken into account.
3 At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate Q12 until the stud is completed.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.
6 Finally, the tool retracts in the tool axis to the clearance height.



Please note while programming:

This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

In the first NC block of the contour program, always program both cylinder surface coordinates.

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

This cycle requires a center-cut end mill (ISO 1641).

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

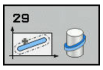
The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.

The set-up clearance must be greater than the tool radius.

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

By setting parameter **CfgGeoCycle** (no. 201000), **displaySpindleErr** (no. 201002) to on/off, you can define whether the control will display an error message (on) or not (off), if the spindle is not active while the cycle is being called. The function needs to be adapted by your machine manufacturer.

Cycle parameters



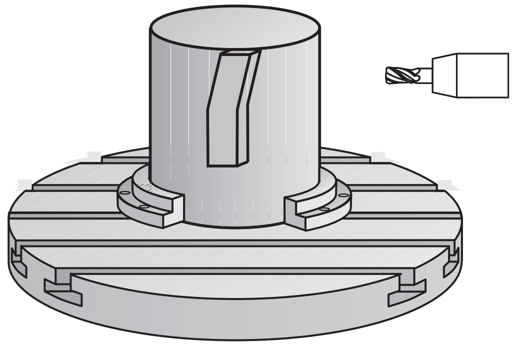
- **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. Input range: -99999.9999 to 99999.9999
- **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999
- **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches
- **Q20 Ridge width?**: Width of the ridge to be machined. Input range: -99999.9999 to 99999.9999

Example

63 CYCL DEF 29 CYL SURFACE RIDGE	
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE F. ROUGHNG
Q16=25	;RADIUS
Q17=0	;TYPE OF DIMENSION
Q20=12	;RIDGE WIDTH

9.5 CYLINDER SURFACE CONTOUR (Cycle 39, ISO: G139, software option 1)

Cycle run



This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the developed surface of the cylinder. With this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the milled contour are always parallel to the cylinder axis.

The contour is described in a subprogram identified in Cycle 14 CONTOUR.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

Unlike in Cycles 28 and 29, in the contour subprogram you define the actual contour to be machined.

1 The control positions the tool above the starting point of machining. The control locates the starting point next to the first point defined in the contour subprogram offset by the tool diameter
2 The control then moves the tool to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate Q12. A finishing allowance programmed for the side is taken into account. The approaching behavior depends on the ConfigDatum, CfgGeoCycle (no. 201000), apprDepCylWall (no. 201004) parameter
3 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12 until the contour train is complete.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth Q1 is reached.
6 Finally, the tool retracts in the tool axis to the clearance height.

Please note while programming:

This cycle performs an inclined machining operation.
To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

In the first NC block of the contour program, always program both cylinder surface coordinates.

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

Ensure that the tool has enough space laterally for contour approach and departure.

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

The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

The set-up clearance must be greater than the tool radius.

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

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

Define the approaching behavior in **ConfigDatum**, **CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004)

- CircleTangential:
  Tangential approach and departure
- LineNormal: The movement to the contour starting point is not performed on a tangential path, but on a straight line

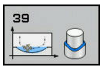
NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** parameter (no. 201002), to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.
- The function needs to be adapted by your machine manufacturer.

Cycle parameters



- **Q1 Milling depth?** (incremental): Distance between workpiece surface and contour bottom. Input range: -99999.9999 to 99999.9999
- **Q3 Finishing allowance for side?** (incremental): Finishing allowance in the plane of the developed cylindrical surface. This allowance is effective in the direction of the radius compensation. Input range: -99999.9999 to 99999.9999
- **Q6 Set-up clearance?** (incremental): Distance between tool tip and cylindrical surface. Input range: 0 to 99999.9999
- **Q10 Plunging depth?** (incremental): Infeed per cut. Input range: -99999.9999 to 99999.9999
- **Q11 Feed rate for plunging?**: Traversing speed of the tool in the spindle axis. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q12 Feed rate for roughing?**: Traversing speed of the tool in the working plane. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q16 Cylinder radius?**: Radius of the cylinder on which the contour is to be machined. Input range: 0 to 99999.9999
- **Q17 Dimension type? deg=0 MM/INCH=1**: Program the coordinates for the rotary axis in the subprogram either in degrees or in mm/inches

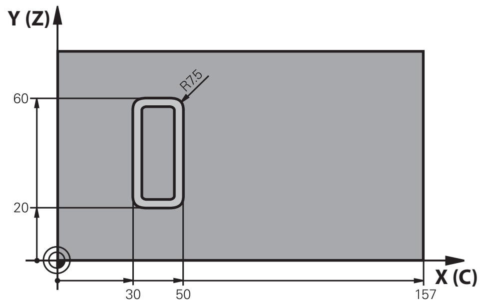
Example

```
63 CYCL DEF 39  CYL. SURFACE
   CONTOUR
   Q1=-8        ;MILLING DEPTH
   Q3=+0        ;ALLOWANCE FOR SIDE
   Q6=+0        ;SET-UP CLEARANCE
   Q10=+3       ;PLUNGING DEPTH
   Q11=100      ;FEED RATE FOR PLNGNG
   Q12=350      ;FEED RATE F. ROUGHNG
   Q16=25       ;RADIUS
   Q17=0        ;TYPE OF DIMENSION
```

9.6 Programming Examples

Example: Cylinder surface with Cycle 27

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

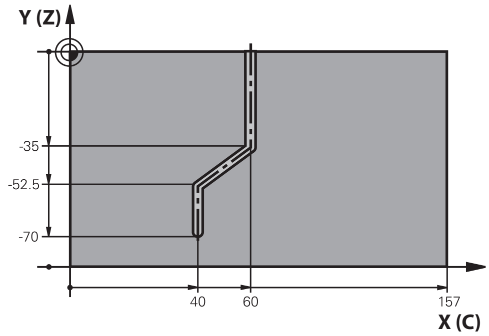


0 BEGIN PGM C27 MM	
1 TOOL CALL 1 Z S2000	Tool call, diameter 7
2 L Z+250 R0 FMAX	Retract the tool
3 L X+50 Y0 R0 FMAX	Pre-position the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MBMAX FMAX	Positioning
5 CYCL DEF 14.0 CONTOUR	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 27 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE F. ROUGHNG	
Q16=25 ;RADIUS	
Q17=1 ;TYPE OF DIMENSION	
8 L C+0 R0 FMAX M13 M99	Pre-position rotary table, spindle ON, call the cycle
9 L Z+250 R0 FMAX	Retract the tool
10 PLANE RESET TURN FMAX	Tilt back, cancel the PLANE function
11 M2	End of program
12 LBL 1	Contour subprogram
13 L X+40 Y+20 RL	Data for the rotary axis are entered in mm (Q17=1)
14 L X+50	
15 RND R7.5	
16 L Y+60	
17 RN R7.5	
18 L IX-20	
19 RND R7.5	

20 L Y+20	
21 RND R7.5	
22 L X+40 Y+20	
23 LBL 0	
24 END PGM C27 MM	

Example: Cylinder surface with Cycle 28

- 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	Define the contour subprogram
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 28 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=-4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE F. ROUGHNG	
Q16=25 ;RADIUS	
Q17=1 ;TYPE OF DIMENSION	
Q20=10 ;SLOT WIDTH	
Q21=0.02 ;TOLERANCE	Remachining active
8 L C+0 R0 FMAX M3 M99	Pre-position rotary table, spindle ON, call the cycle
9 L Z+250 R0 FMAX	Retract the tool
10 PLANE RESET TURN FMAX	Tilt back, cancel the PLANE function
11 M2	End of program
12 LBL 1	Contour subprogram, description of the center path
13 L X+60 Y+0 RL	Data for the rotary axis are entered in mm (Q17=1)
14 L Y-35	
15 L X+40 Y-52.5	
16 L Y-70	
17 LBL 0	
18 END PGM C28 MM	

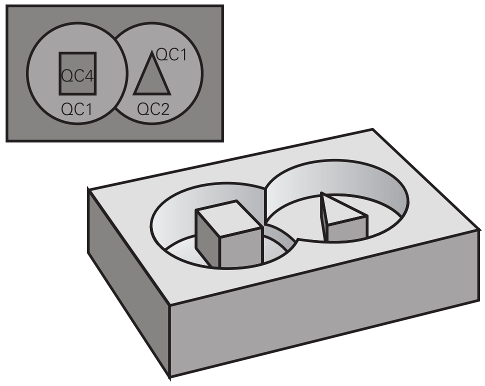
10

Fixed Cycles: Contour Pocket with Contour Formula

10.1 SL cycles with complex contour formula

Fundamentals

Using SL cycles and the complex contour formula, you can combine subcontours (pockets or islands) to form complex contours. You define the individual subcontours (geometry data) in separate NC programs. In this way, any subcontour can be reused any number of times. The control calculates the complete contour from the selected subcontours, which you link through a contour formula.



> The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** elements.
>
> To use SL cycles with contour formulas, it is mandatory that your program is structured carefully. These cycles enable you to save frequently used contours in individual NC programs. Using the contour formula, you can connect the subcontours to define a complete contour and specify whether it applies to a pocket or island.
>
> In its present form, the "SL cycles with contour formula" function requires input from several areas in the control user interface. This function will serve as a basis for further development.

Program structure: Machining with SL cycles and complex contour formula

0 BEGIN PGM CONTOUR MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 CONTOUR DATA ...
8 CYCL DEF 22 ROUGHING ...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING ...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING ...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTOUR MM

Properties of the subcontours

- The control assumes that each contour is a pocket. Do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram.
- You can define subcontours with various depths as needed

Characteristics of the fixed cycles

- The control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions—the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Program structure: Calculation of the subcontours with contour formula

```
0 BEGIN PGM MODEL MM
1 DECLARE CONTOUR QC1 = "CIRCLE1"
2 DECLARE CONTOUR QC2 =
    "CIRCLEXY" DEPTH15
3 DECLARE CONTOUR QC3 =
    "TRIANGLE" DEPTH10
4 DECLARE CONTOUR QC4 = "SQUARE"
    DEPTH5
5 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
6 END PGM MODEL MM
```

```
0 BEGIN PGM CIRCLE1 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM CIRCLE1 MM
```

```
0 BEGIN PGM CIRCLE31XY MM
...
...
```

Selecting an NC program with contour definitions

With the **SEL CONTOUR** function, you select an NC program with contour NC, from which the control extracts the contour descriptions:

SPEC FCT
- Show the soft-key row with special functions

CONTOUR + POINT MACHINING
- Menu for functions: Press the soft key for contour and point machining

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

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

Defining contour descriptions

Using the **DECLARE CONTOUR** function in your NC program, you enter the path for NC programs from which the control extracts the contour descriptions. In addition, you can select a separate depth for this contour description (FCL 2 function):

SPEC FCT
- Show the soft-key row with special functions

CONTOUR + POINT MACHINING
- Menu for functions: Press the soft key for contour and point machining

DECLARE CONTOUR
- Press the **DECLARE CONTOUR** soft key.
- Enter the number for the contour designator **QC**, and confirm with the **ENT** key
- Enter the full name of the NC program with the contour descriptions and confirm with the **END** key, or if desired,
- Define a separate depth for the selected contour

> With the entered contour designators **QC** you can include the various contours in the contour formula.
>
> If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).

Entering a complex contour formula

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

SPEC FCT

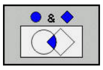
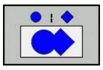
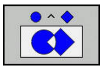
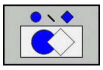
- Show the soft-key row with special functions

CONTOUR + POINT MACHINING

- Menu for functions: Press the soft key for contour and point machining

CONTOUR FORMULA

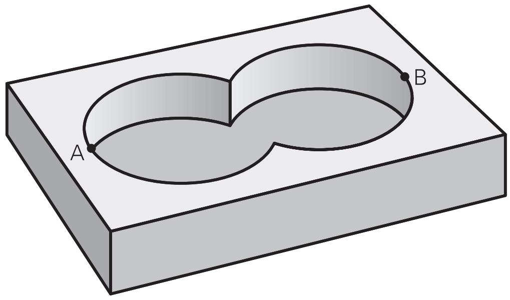
- Press the **CONTOUR FORMULA** soft key. The control then displays the following soft keys:

Soft key	Mathematical function
	**Intersected with** e.g. **QC10= QC1& QC5**
	**Joined with** e.g. **QC25= QC7| QC18**
	**joined with, but without intersection** e.g. **QC12 = QC5 ^ QC25**
	**without** e.g. **QC25 = QC1 \ QC2**
(	**Open parenthesis** e.g. **QC12 = QC1 * (QC2 + QC3)**
)	**Close parenthesis** e.g. **QC12 = QC1 * (QC2 + QC3)**
	Define single contour z. B. **QC12 = QC1**

Superimposed contours

By default, the control considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.



Subprograms: overlapping pockets

> 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

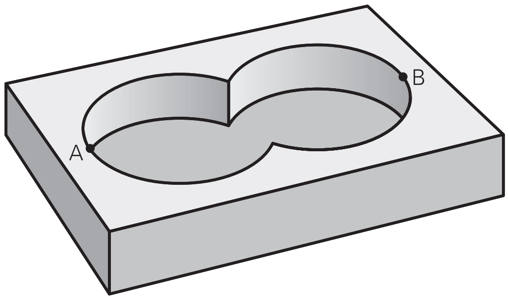
Contour description program 2: pocket B

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.



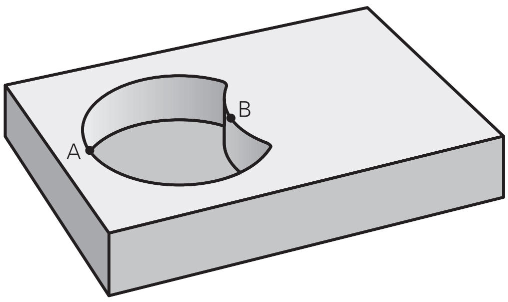
Contour definition program:

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.



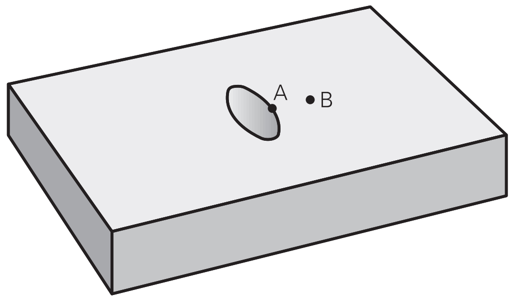
Contour definition program:

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.



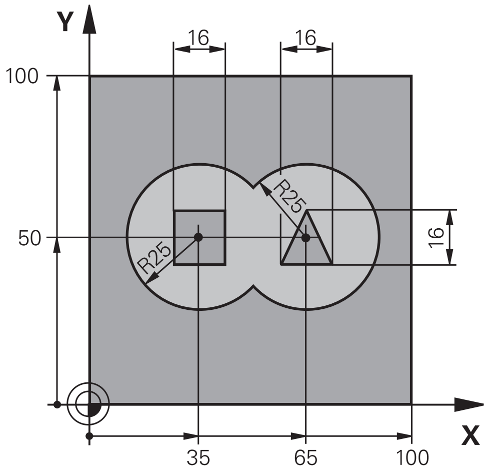
Contour definition program:

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 230).

Example: Roughing and finishing superimposed contours with the contour formula



0 BEGIN PGM CONTOUR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S2500	Tool call: roughing cutter
4 L Z+250 R0 FMAX	Retract the tool
5 SEL CONTOUR “MODEL“	Define contour definition program
6 CYCL DEF 20 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;ROTATIONAL DIRECTION	

7 CYCL DEF 22 ROUGH-OUT	Cycle definition: rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE F. ROUGHNG	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;FEED RATE FOR RECIP.	
Q401=100 ;FEED RATE FACTOR	
Q404=0 ;FINE ROUGH STRATEGY	
8 CYCL CALL M3	Cycle call: rough-out
9 TOOL CALL 2 Z S5000	Tool call: finishing cutter
10 CYCL DEF 23 FLOOR FINISHING	Cycle definition: floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE F. ROUGHNG	
11 CYCL CALL M3	Cycle call: floor finishing
12 CYCL DEF 24 SIDE FINISHING	Cycle definition: side finishing
Q9=+1 ;ROTATIONAL DIRECTION	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE F. ROUGHNG	
Q14=+0 ;ALLOWANCE FOR SIDE	
13 CYCL CALL M3	Cycle call: side finishing
14 L Z+250 R0 FMAX M2	Retract the tool, end program
15 END PGM KONTUR MM	

Contour definition program with contour formula:

0 BEGIN PGM MODEL MM	Contour definition program
1 DECLARE CONTOUR QC1 = “CIRCLE1”	Definition of the contour label for the "CIRCLE1" NC program
2 FN 0: Q1 =+35	Assignment of values for parameters used in PGM "CIRCLE31XY"
3 FN 0: Q2 =+50	
4 FN 0: Q3 =+25	
5 DECLARE CONTOUR QC2 = “CIRCLE31XY”	Definition of the contour label for the "CIRCLE3XY" NC program
6 DECLARE CONTOUR QC3 = “TRIANGLE”	Definition of the contour label for the "TRIANGLE" NC program
7 DECLARE CONTOUR QC4 = “SQUARE”	Definition of the contour label for the "SQUARE" NC program
8 QC10 = (QC 1 | QC 2) \ QC 3 \ QC 4	Contour formula
9 END PGM MODEL MM	

Contour description programs:

0 BEGIN PGM CIRCLE1 MM	Contour description program: circle at right
1 CC X+65 Y+50	
2 L PR+25 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE1 MM	

0 BEGIN PGM CIRCLE31XY MM	Contour description program: circle at left
1 CC X+Q1 Y+Q2	
2 LP PR+Q3 PA+0 R0	
3 CP IPA+360 DR+	
4 END PGM CIRCLE31XY MM	

0 BEGIN PGM TRIANGLE MM	Contour description program: triangle at right
1 L X+73 Y+42 R0	
2 L X+65 Y+58	
3 L X+58 Y+42	
4 L X+73	
5 END PGM TRIANGLE MM	

0 BEGIN PGM SQUARE MM	Contour description program: square at left
1 L X+27 Y+58 R0	
2 L X+43	
3 L Y+42	
4 L X+27	
5 L Y+58	
6 END PGM SQUARE MM	

10.2 SL cycles with simple contour formula

Fundamentals

Using SL cycles and the simple contour formula, you can easily combine up to nine subcontours (pockets or islands) to form a particular contour. You define the individual subcontours (geometry data) in separate NC programs. In this way, any subcontour can be reused any number of times. The control calculates the complete contour from the selected subcontours.

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

Program structure: Machining with SL cycles and complex contour formula

```
0 BEGIN PGM CONTDEF MM
...
5 CONTOUR DEF P1= “POCK1.H“ I2 =
    “ISLE2.H“ DEPTH5 I3 “ISLE3.H“
    DEPTH7.5
6 CYCL DEF 20 CONTOUR DATA ...
8 CYCL DEF 22 ROUGHING ...
9 CYCL CALL
...
12 CYCL DEF 23 FLOOR FINISHING ...
13 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING ...
17 CYCL CALL
63 L Z+250 R0 FMAX M2
64 END PGM CONTDEF MM
```

Properties of the subcontours

- Do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram.

Characteristics of the fixed cycles

- The control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions—the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Entering a simple contour formula

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

SPEC FCT

- Show the soft-key row with special functions

CONTOUR + POINT MACHINING

- Menu for functions: Press the soft key for contour and point machining

CONTOUR DEF

- Press the **CONTOUR DEF** soft key. The control opens the dialog for entering the contour formula.
- Enter the name of the first subcontour. The first subcontour must always be the deepest pocket. Confirm with the **ENT** key.

ISLAND

- Specify via soft key whether the next subcontour is a pocket or an island. Confirm with the **ENT** key.
- Enter the name of the second subcontour. Confirm with the **ENT** key
- If needed, enter the depth of the second subcontour. Confirm with the **ENT** key.
- Carry on with the dialog as described above until you have entered all subcontours.

> 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 230).

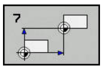
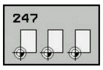
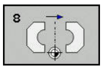
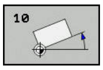
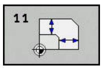
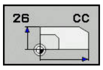
11

Cycles: Coordinate Transformations

11.1 Fundamentals

Overview

Once a contour has been programmed, the control can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The control provides the following functions for coordinate transformations:

Soft key	Cycle	Page
	7 DATUM SHIFT For shifting contours directly within the NC program or from datum tables	305
	247 PRESETTING Presetting during the program run	311
	8 MIRRORING Mirroring contours	312
	10 ROTATION Rotating contours in the working plane	314
	11 SCALING FACTOR Increasing or reducing the size of contours	316
	26 AXIS-SPECIFIC SCALING Increasing or reducing the size of contours with axis-specific scaling	317
	19 WORKING PLANE Machining in a tilted coordinate system on machines with swivel heads and/or rotary tables	319

Effectiveness of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

Reset coordinate transformation:

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

11.2 DATUM SHIFT (Cycle 7, ISO: G54)

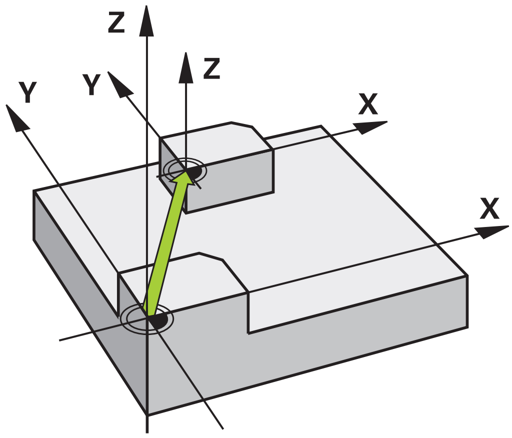
Effect

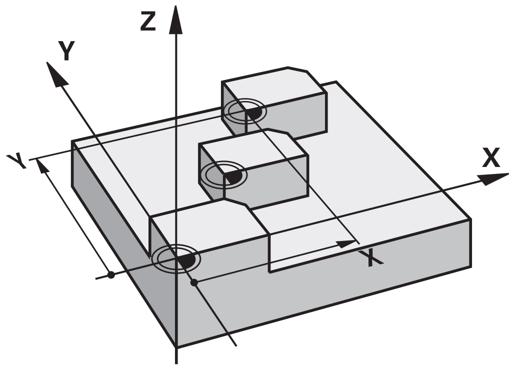
A datum shift allows machining operations to be repeated at various locations on the workpiece.

After the definition of a datum shift cycle, all coordinate data will reference the new datum. The control displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

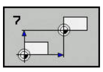
Resetting

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





Cycle parameters



- **Displacement**: Enter the coordinates of the new datum. Absolute values reference the workpiece datum defined by presetting. Incremental values always reference the datum which was last valid—this can be a datum which has already been shifted. Input range: Up to six NC axes, each from –99999.9999 to 99999.9999

Example

```
13 CYCL DEF 7.0  DATUM SHIFT
14 CYCL DEF 7.1  X+60
15 CYCL DEF 7.2  Y+40
16 CYCL DEF 7.3  Z-5
```

Please note while programming

Refer to your machine manual!

Possible datum shift values in the rotary axes will be specified by your machine tool builder in the **presetToAlignAxis** parameter (no. 300203).

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

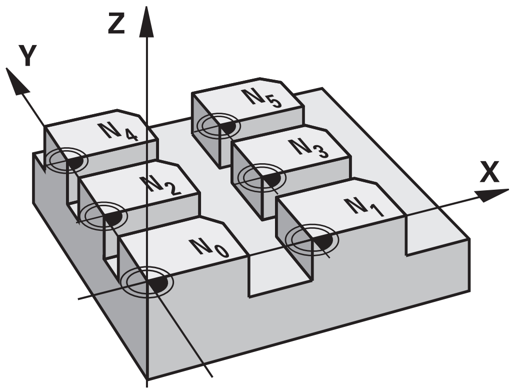
11.3 DATUM SHIFT with datum tables (Cycle 7, ISO: G53)

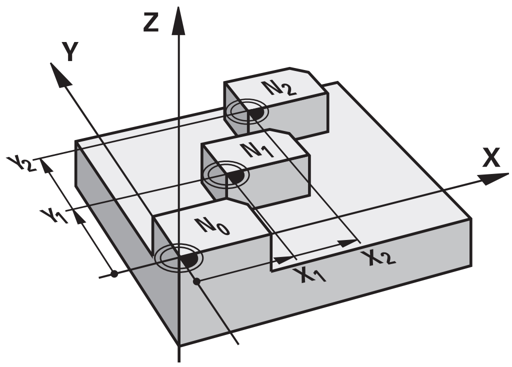
Effect

Datum tables are used for:

- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within an NC program, you can either program datum points directly in the cycle definition or call them from a datum table.





Resetting

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

Status displays

In the additional status display, the following data from the datum table is shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

Please note while programming:

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

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

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

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

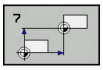
- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table now has the status S
- Use the file management to select the desired table for the **Program run, single block** and **Program run, full sequence** operating modes: The table now has the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

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

New lines can only be inserted at the end of the table.

If you create datum tables, the file name has to start with a letter.

Cycle parameters



- **Displacement**: Enter the number of the datum from the datum table or in a Q parameter. If you enter a Q parameter, the control activates the datum number entered in the Q parameter. Input range: 0 to 9999

Example

```
77 CYCL DEF 7.0  DATUM SHIFT
78 CYCL DEF 7.1  #5
```

Selecting a datum table in the part program

With the **SEL TABLE** function, you select the datum table from which the control takes the datums:

PGM CALL

- To select the functions for program call, press the **PGM CALL** key

DATUM TABLE

- Press the **DATUM TABLE** soft key
- Enter the complete path name of the datum table or select the file with the **SELECT** soft key. Confirm your input with the **END** key.

> Program a **SEL TABLE** block before Cycle 7 Datum Shift.
>
> A datum table selected with **SEL TABLE** remains active until you select another datum table with **SEL TABLE** or through **PGM MGT**.

Editing the datum table in the Programming mode of operation

> After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise, the change will not be taken into account when the NC program is executed.

Select the datum table in the **Programming** mode of operation**Programming**

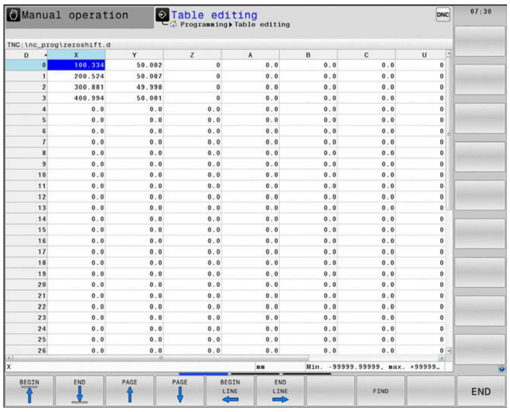
PGM MGT

- To call the file manager, press the **PGM MGT** key.
- Display the datum tables: Press the **SELECT TYPE** and **SHOW .D** soft keys
- Select the desired table or enter a new file name.
- Edit the file. The functions in the soft-key row include:

Soft key	Function
BEGIN ↑	Select the beginning of the table
END ↓	Select the table end
PAGE ↑	Go to previous page
PAGE ↓	Go to next page
INSERT LINE	Insert line (only possible at the end of table)
DELETE LINE	Delete line
FIND	Find
BEGIN LINE ←	Move the cursor to the beginning of the line
END LINE →	Move the cursor to the end of the line
COPY FIELD	Copy the current value
PASTE FIELD	Insert the copied value
APPEND N LINES AT END	Add the entered number of lines (datums) to the end of the table

Configuring a datum table

If you do not wish to define a datum for an active axis, press the **DEL** key. Then the control clears the numerical value from the corresponding input field.



> You can change the properties of tables. Enter code number 555343 in the MOD menu. The control then displays the **EDIT FORMAT** soft key if a table is selected. When you press this soft key, the control opens a pop-up window where the properties are shown for each column of the selected table. Any changes you make only affect the open table.

Leaving a datum table

Select a different type of file in file management. Select the desired file.

NOTICE

Danger of collision!

The control considers changes in a datum table only when the values are saved.

- Make sure to confirm any changes made to the table immediately by pressing the **ENT** key
- Carefully test the NC program after making a change to the datum table

Status displays

In the additional status display, the control shows the values of the active datum shift.

11.4 PRESETTING (Cycle 247, ISO: G247)

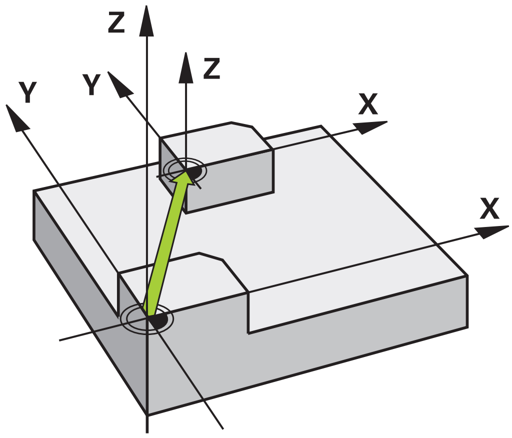
Effect

With the presetting cycle you can activate as the new preset a preset defined in the preset table.

After a presetting cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

Status display

In the status display; the control shows the active preset number behind the preset symbol.



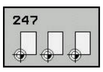
Please note before programming:

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 behind the **Preset** dialog.

11.5 MIRRORING (Cycle 8, ISO: G28)

Effect

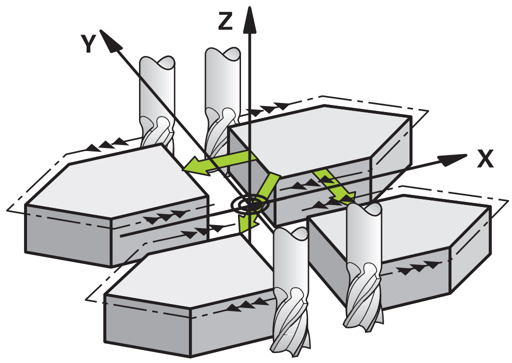
The control can machine the mirror image of a contour in the working plane.

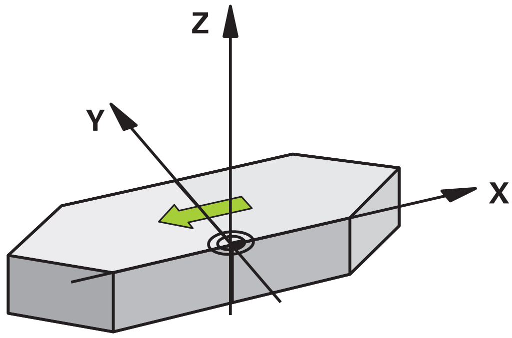
Mirroring becomes effective as soon as it has been defined in the NC program. It is also effective in the **Positioning w/ Manual Data Input** operating mode. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed (except in SL cycles).
- If you mirror two axes, the machining direction remains the same.

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

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





Resetting

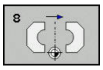
Program the MIRROR IMAGE cycle once again with **NO ENT**.

Please note while programming:

If you work in a tilted system with Cycle 8 the following procedure is recommended:

- **First** program the tilting movement and **then** call Cycle 8 MIRRORING!

Cycle parameters



- **Mirror image axis?**: Enter the axis to be mirrored. You can mirror all axes—including rotary axes—except for the spindle axis and its associated secondary axis. You can enter up to three axes. Input range: up to three NC axes **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example

```
79 CYCL DEF 8.0  MIRRORING
80 CYCL DEF 8.1  X Y  Z
```

11.6 ROTATION (Cycle 10, ISO: G73)

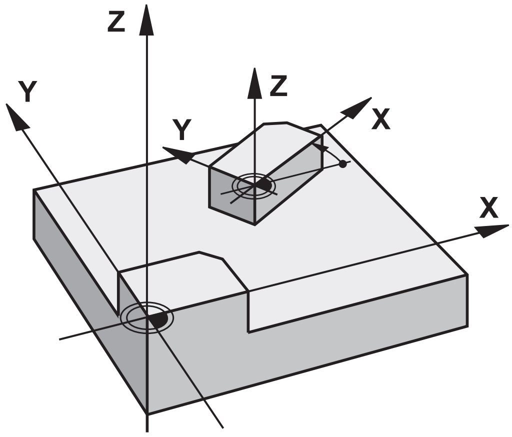
Effect

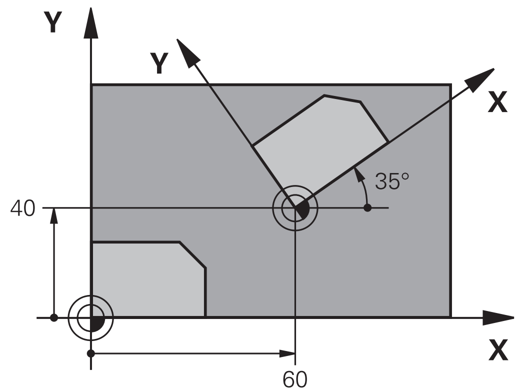
Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

The ROTATION cycle becomes effective as soon as it has been defined in the NC program. It is also effective in the Positioning with Manual Data Input operating mode. The active angle of rotation is shown in the additional status display.

Reference axis for the rotation angle:

- X/Y plane: X axis
- Y/Z plane: Y axis
- Z/X plane: Z axis





Resetting

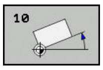
Program the ROTATION cycle once again with a rotation angle of 0°.

Please note while programming:

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

After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

Cycle parameters



- **Rotation**: Enter the angle of rotation in degrees (°). Input range: -360.000° to +360.000° (absolute or incremental)

Example

```
12 CALL LBL 1
13 CYCL DEF 7.0  DATUM SHIFT
14 CYCL DEF 7.1  X+60
15 CYCL DEF 7.2  Y+40
16 CYCL DEF 10.0  ROTATION
17 CYCL DEF 10.1  ROT+35
18 CALL LBL 1
```

11.7 SCALING (Cycle 11, ISO: G72)

Effect

The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

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

The scaling factor has an effect on

- all three coordinate axes at the same time
- dimensions in cycles

Prerequisite

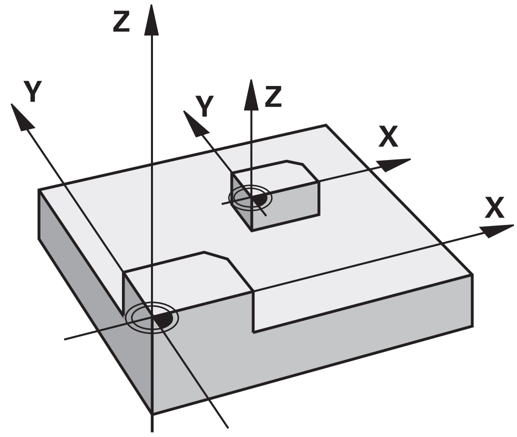
It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

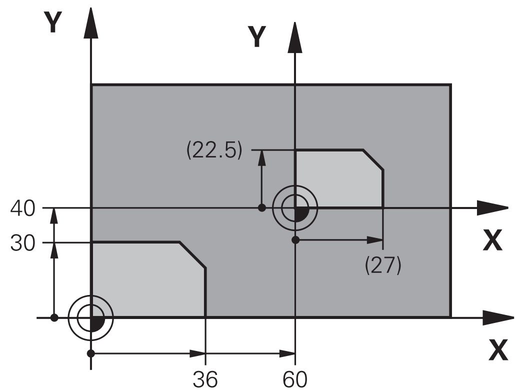
Enlargement: SCL greater than 1 (up to 99.999 999)

Reduction: SCL less than 1 (down to 0.000 001)

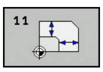
Resetting

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





Cycle parameters



- **Factor?**: Enter the scaling factor SCL. The control multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range: 0.000001 to 99.999999

Example

```
11 CALL LBL 1
12 CYCL DEF 7.0  DATUM SHIFT
13 CYCL DEF 7.1  X+60
14 CYCL DEF 7.2  Y+40
15 CYCL DEF 11.0  SCALING
16 CYCL DEF 11.1  SCL 0.75
17 CALL LBL 1
```

11.8 AXIS-SPECIFIC SCALING (Cycle 26)

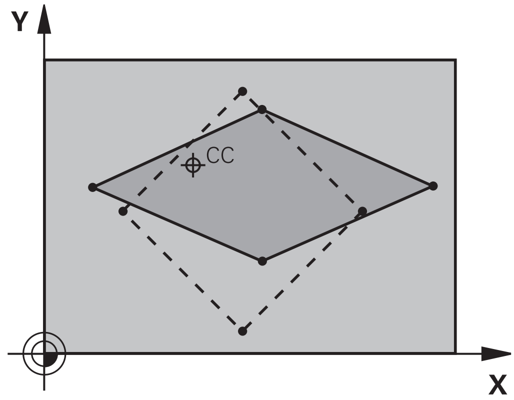
Effect

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

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

Resetting

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



Please note while programming:

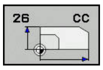
Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

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

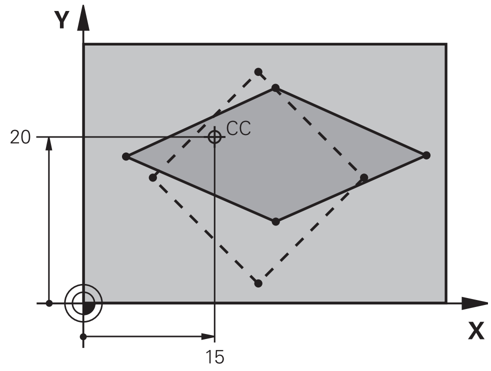
In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.

Cycle parameters



- **Axis and factor**: Select the coordinate axis/axes via soft key. Enter the factor(s) for axis-specific enlargement or reduction. Input range: 0.000001 to 99.999999
- **Center coordinates**: Enter the center of the axis-specific enlargement or reduction. Input range -99999.9999 to 99999.9999



Example

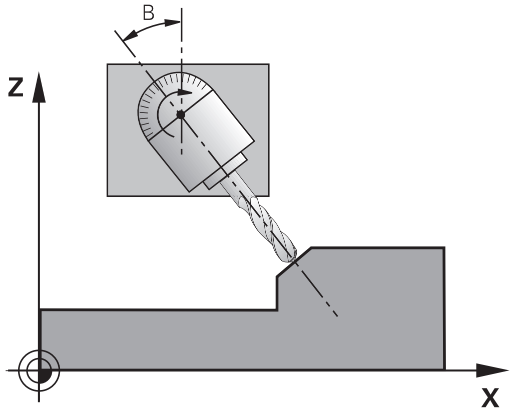
25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL 1

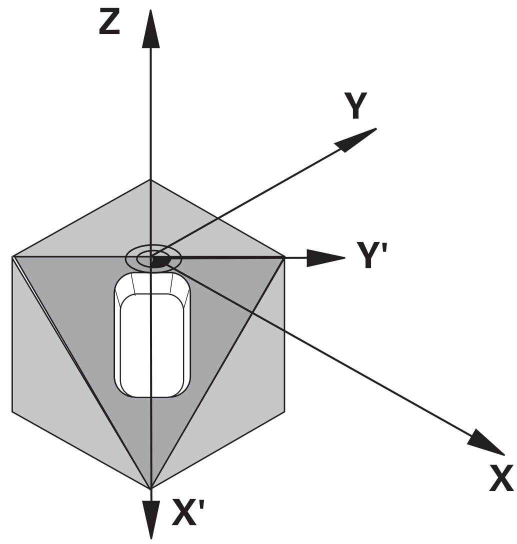
11.9 WORKING PLANE (Cycle 19, ISO: G80, software option 1)

Effect

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

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to three rotations (spatial angles) of the **machine-based** coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.





> Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.

If you program the position of the working plane via spatial angles, the control will calculate the required angle positions of the tilted axes automatically and will store these in the Q120 (A axis) to Q122 (C axis) parameters. If two solutions are possible, the control will choose the shorter path from the current position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The control first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it has been defined in the NC program. As soon as you move an axis in the tilted system, the compensation for this specific axis will be activated. You must move all axes to activate compensation for all axes.

If you set the **Tilting program run** parameters to **Active** in the Manual Operation mode, the angular value entered in this menu is overwritten by Cycle 19 WORKING PLANE.

Please note while programming:

The **Tilt working plane** functions are interfaced to the control and the machine tool by the machine tool builder.

The machine tool builder also specifies whether the programmed angles are interpreted as coordinates of the rotary axes (axis angles) or as angular components of a tilted plane (spatial angles).

Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

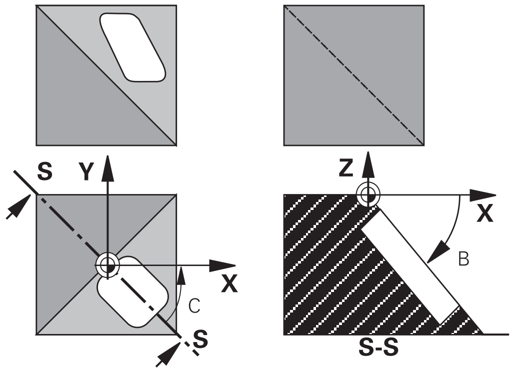
The working plane is always tilted around the active datum.

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

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

Cycle parameters

- **Rotary axis and angle?**: Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys. Input range: -360.000 to 360.000



If the control automatically positions the rotary axes, you can enter the following parameters:

- **Feed rate? F=**: Traversing speed of the rotary axis during automatic positioning. Input range 0 to 99999.999
- **Set-up clearance?** (incremental): The control positions the tilting head in such a way that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. Input range: 0 to 99999.9999

Reset

To reset the tilt angles, redefine the WORKING PLANE cycle. Enter an angular value of 0° for all rotary axes. Then, redefine the WORKING PLANE cycle. Confirm the dialog prompt by pressing the **NO ENT** key. This disables the function.

Positioning the axes of rotation

Refer to your machine manual.

The machine tool builder determines whether Cycle 19 positions the axes of rotation automatically or whether they must be positioned manually in the NC program.

Manual positioning of rotary axes

If Cycle 19 does not position the rotary axes automatically, you need to position them in a separate L block following the cycle definition.

If you use axis angles, you can define the axis values right in the L block. If you use spatial angles, then program the Q parameters **Q120** (A axis value), **Q121** (B axis value) and **Q122** (C axis value) according to Cycle 19.

For manual positioning, always use the rotary axis positions stored in Q parameters Q120 to Q122.

Avoid using functions, such as M94 (modulo rotary axes), in order to avoid discrepancies between the actual and nominal positions of rotary axes in multiple definitions.

Example

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the spatial angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0	
14 L A+Q120 C+Q122 R0 F1000	Position the rotary axes by using values calculated by Cycle 19
15 L Z+80 R0 FMAX	Activate compensation for the spindle axis
16 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle 19:

- The control can position only closed-loop axes.
- To position the tilted axes, you must enter a feed rate and a set-up clearance, in addition to the tilting angles, when defining the cycle.
- Use only preset tools (the full tool length must be defined).
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting
- The control performs tilting at the last programmed feed rate. The maximum feed rate depends on the complexity of the swivel head (tilting table).

Example

10 L Z+100 R0 FMAX	
11 L X+25 Y+10 R0 FMAX	
12 CYCL DEF 19.0 WORKING PLANE	Define the angle for calculation of the compensation
13 CYCL DEF 19.1 A+0 B+45 C+0 F5000 ABST50	Also define the feed rate and the clearance
14 L Z+80 R0 FMAX	Activate compensation for the spindle axis
15 L X-8.5 Y-10 R0 FMAX	Activate compensation for the working plane

Position display in a tilted system

On activation of Cycle 19, the displayed positions (**ACTL** and **NOML**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle 19.

Monitoring of the working space

The control monitors only those axes in the tilted coordinate system that are moved. Where applicable, the control displays an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system.

It is also possible to position the axes using straight-line blocks that reference the machine coordinate system (NC blocks with M91 or M92), if the working plane is tilted. Restrictions:

- Positioning is without length compensation.
- Positioning is done without length compensation.
- Tool radius compensation is not allowed.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is tilted about the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after having activated Cycle 19, you are shifting the tilted coordinate system.

Important: When resetting the cycles, use the reverse sequence used for defining them:

1st Activate datum shift
2nd Activate tilting function
3rd Activate rotation
...
Workpiece machining
...
1st Reset rotation
2nd Reset tilting function
3rd Reset datum shift

Procedure for working with Cycle 19 WORKING PLANE

1 Create the NC program

- Define the tool (not required if TOOL.T is active), and enter the full tool length.
- Call the tool.
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece or clamping devices during tilting.
- If required, position the tilt axis or axes with an L block to the appropriate angular value(s) (depending on a machine parameter).
- Activate datum shift if required.
- Define Cycle 19 WORKING PLANE; enter the angular values for the tilt axes.
- Traverse all principal axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- If required, define Cycle 19 WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle 19—you can define the new angular values directly.
- Reset Cycle 19 WORKING PLANE by programming 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle 19. Confirm the dialog prompt with **NO ENT**.
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

2 Clamp the workpiece

3 Presetting

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

  Further information:: User's Manual for Setup, Testing and Running NC programs
- Automatically with a HEIDENHAIN 3-D touch probe
  Further information: "Touch Probe Cycles: Automatic Presetting", Page 589)

4 Start the NC program in the Program Run, Full Sequence operating mode

5 Manual Operation mode

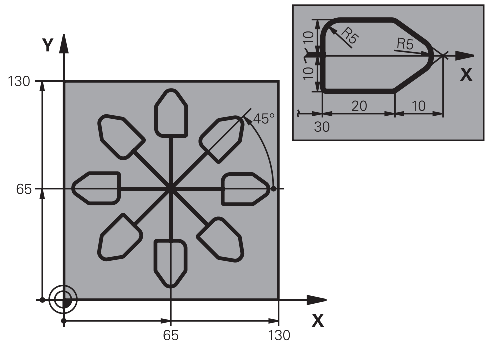
Use the 3-D ROT soft key to set the TILT WORKING PLANE function to INACTIVE. Enter an angular value of 0° for each rotary axis in the menu.

11.10 Programming Examples

Example: coordinate transformation cycles

Program run

- Program the coordinate transformations in the main program
- Machining within a subprogram



0 BEGIN PGM COTRANS MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Workpiece blank definition
2 BLK FORM 0.2 X+130 X+130 Z+0	
3 TOOL CALL 1 Z S4500	Tool call
4 L Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center
6 CYCL DEF 7.1 X+65	
7 CYCL DEF 7.2 Y+65	
8 CALL LBL 1	Call milling operation
9 LBL 10	Set label for program section repeat
10 CYCL DEF 10.0 ROTATION	Rotate by 45° (incremental)
11 CYCL DEF 10.1 IROT+45	
12 CALL LBL 1	Call milling operation
13 CALL LBL 10 REP 6/6	Return jump to LBL 10; repeat the milling operation six times
14 CYCL DEF 10.0 ROTATION	Reset the rotation
15 CYCL DEF 10.1 ROT+0	
16 CYCL DEF 7.0 DATUM SHIFT	Reset datum shift
17 CYCL DEF 7.1 X+0	
18 CYCL DEF 7.2 Y+0	
19 L Z+250 R0 FMAX M2	Retract the tool, end program
20 LBL 1	Subprogram 1
21 L X+0 Y+0 R0 FMAX	Define milling operation
22 L Z+2 R0 FMAX M3	
23 L Z-5 R0 F200	
24 L X+30 RL	
25 L IY+10	
26 RND R5	
27 L IX+20	
28 L IX+10 IY-10	

29 RND R5	
30 L IX-10 IY-10	
31 L IX-20	
32 L IY+10	
33 L X+0 Y+0 R0 F5000	
34 L Z+20 R0 FMAX	
35 LBL 0	
36 END PGM COTRANS MM	

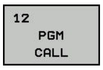
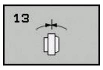
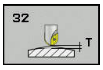
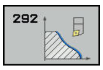
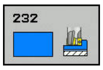
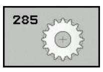
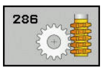
12

Cycles: Special Functions

12.1 Fundamentals

Overview

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

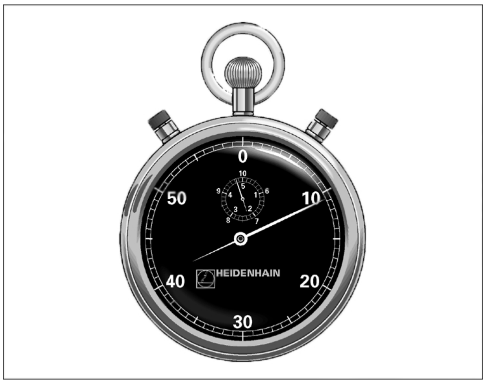
Soft key	Cycle	Page
	9 DWELL TIME	331
	12 Program call	332
	13 Oriented spindle stop	333
	32 TOLERANCE	334
	225 ENGRAVING of texts	358
	291 COUPLING TURNING INTERPOLATION	350
	292 CONTOUR TURNING INTERPOLATION	338
	232 FACE MILLING	364
	239 ASCERTAIN THE LOAD	369
	285 DEFINE GEAR	376
	286 GEAR HOBBING	379
	287 GEAR SKIVING	385

12.2 DWELL TIME (Cycle 9, ISO: G04)

Function

Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.

The cycle becomes effective as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.



Example

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

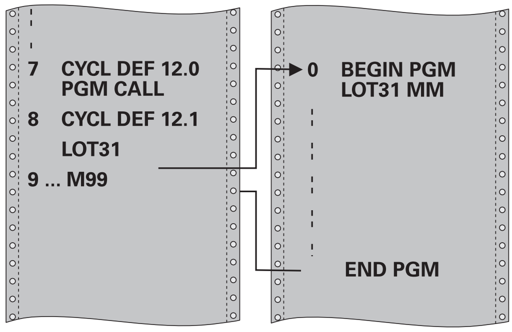
Cycle parameters

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

12.3 PROGRAM CALL (Cycle 12, ISO: G39)

Cycle function

NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.



Please note while programming:

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

12 PGM CALL

- **Program name**: Enter the name of the NC program and, if necessary, the path where it is located, or
- Activate the file select dialog with the **SELECT** soft key. Select the NC program to be called.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare program 50.h as a cycle and call it with M99

55 CYCL DEF 12.0 PGM CALL
56 CYCL DE 12.1 PGM TNC: \KLAR35\FK1\50.H
57 L X+20 Y+50 FMAX M99

12.4 SPINDLE ORIENTATION (Cycle 13, ISO: G36)

Cycle function

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

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

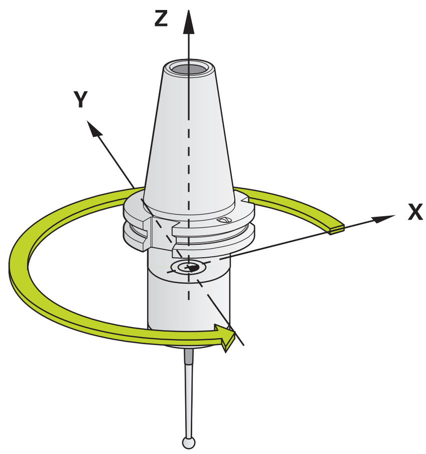
Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

With M19 or M20, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program M19 or M20 without having defined Cycle 13 beforehand, the control positions the main spindle at an angle that has been set by the machine tool builder.

More information: machine tool manual.



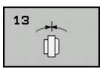
Example

93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180

Please note while programming:

> Cycle 13 is used internally for Cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

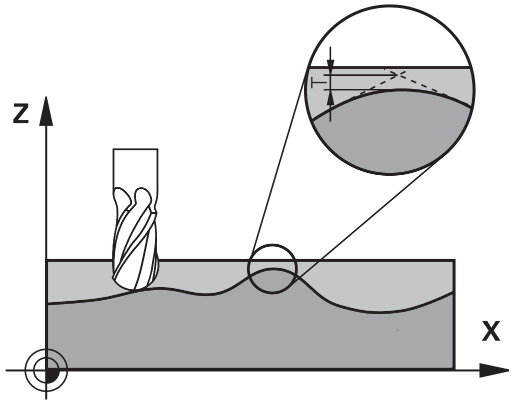


- **Angle of orientation**: Enter the angle with reference to the angle reference axis of the working plane. Input range: 0.0000° to 360.0000°

12.5 TOLERANCE (Cycle 32, ISO: G62)

Cycle function

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



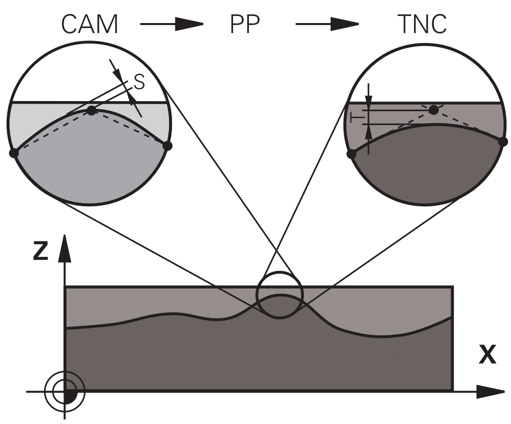
With the entries in Cycle 32 you can influence the result of HSC machining with respect to accuracy, surface definition and speed, inasmuch as the control has been adapted to the machine's characteristics.

The control automatically smooths the contour between any two contour elements (whether compensated or not). This means that the tool has constant contact with the workpiece surface and therefore reduces wear on the machine tool. The tolerance defined in the cycle also affects the traverse paths on circular arcs.

If necessary, the control automatically reduces the programmed feed rate so that the program can be executed at the fastest possible speed without jerking. **Even if the control does not move the axes with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the control can move the axes.

Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error (**tolerance value**) is set in a machine parameter by the machine manufacturer. With **Cycle 32** you can change the pre-set tolerance value and select different filter settings, provided that your machine tool builder has implemented these features.

Influences of the geometry definition in the CAM system



The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system. The chord error defines the maximum point spacing of NC programs generated in a post processor (PP). If the chord error is less than or equal to the tolerance value **T** defined in Cycle 32, then the control can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing if in Cycle 32 you choose a tolerance value between 110-% and 200-% of the CAM chord error.

Please note while programming!

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the control, but by the fact that, in order to machine the contour transitions very exactly, the control might have to drastically reduce the speed.

Cycle 32 is DEF-active which means that it becomes effective as soon as it is defined in the NC program.

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

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

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

In a program with millimeters set as unit of measure, the control interprets the entered tolerance value **T** in millimeters. In an inch program it interprets it as inches.

If you load an NC program with Cycle 32 that contains only the **Tolerance value** T cycle parameter, the control inserts the two remaining parameters with the value 0 if required.

As the tolerance value increases, the diameter of circular movements usually decreases, unless HSC filters are active on your machine (set by the machine tool builder).

If Cycle 32 is active, the control shows the parameters defined for Cycle 32 on the **CYC** tab of the additional status display.

NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more consistent. Additionally, in you can set a higher rotational axis tolerance **TA** (e.g. between 1° and 3°) for an even more constant feed-rate curve at the tool reference point (TCP).

For NC programs for 5-axis simultaneous machining with toroid cutters or spherical cutters, where the NC output is for the south pole of the sphere, choose a lower rotary axis tolerance. 0.1° is a typical value. However, the maximum permissible contour damage is the decisive factor for the rotary axis tolerance. This contour damage depends on the possible tool tilting, tool radius and engagement depth of the tool.
With 5-axis hobbing with an end mill, you can calculate the maximum possible contour damage T directly from the cutter engagement length L and permissible contour tolerance TA:
T ~ K x L x TA K = 0.0175 [1/°]
Example: L = 10 mm, TA = 0.1°: T = 0.0175 mm

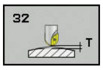
Sample formula for a toroid cutter:

When machining with a toroid cutter, the angle tolerance is very important.

$$Tw = \frac{180}{\Pi * R} T_{32}$$

T_w: Angle tolerance in degrees
π
R: Major radius in mm
T_{32}: Machining tolerance in mm

Cycle parameters



- **Tolerance value T**: Permissible contour deviation in mm (or inches with inch programming). Input range: 0.0000 to 10.0000
  >0: If you enter a value greater than zero, the control will use the maximum permissible deviation you specify
  0: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine tool builder.
- **HSC MODE, Finishing=0, Roughing=1**: Activate filter:
  - Input value 0: **Milling with increased contour accuracy**. The control uses internally defined finishing filter settings.
  - Input value 1: **Milling at an increased feed rate**. The control uses internally defined roughing filter settings.
- **Tolerance for rotary axes TA**: Permissible position error of rotary axes in degrees when M128 is active (FUNCTION TCPM). The control always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for NC programs for more than one axis by entering a large tolerance value (e.g. 10°), because the control does not always have to position the rotary axis at the given nominal position. The tool orientation (position of the rotary axis with respect to the workpiece surface) will be adjusted. The position at the **T**ool **C**enter **P**oint (TCP) will be corrected automatically. For example, with a spherical cutter measured in its center and programmed based on the center path, there will be no adverse effects on the contour. Input range: 0.0000 to 10.0000
  >0: If you enter a value greater than zero, the control will use the maximum permissible deviation you specify.
  0: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine tool builder.

Example

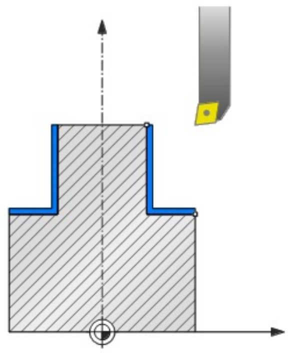
```
95 CYCL DEF 32.0  TOLERANCE
96 CYCL DEF 32.1  T0.05
97 CYCL DEF 32.2  HSC-MODE:1 TA5
```

12.6 INTERPOLATION TURNING, CONTOUR FINISHING (Cycle 292, ISO: G292, software option 96)

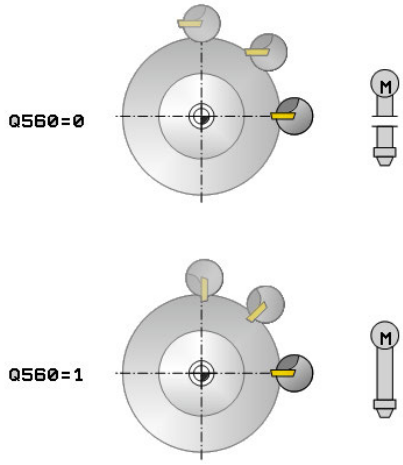
Cycle run

Cycle 292 INTERPOLATION TURNING, CONTOUR FINISHING couples the tool spindle to the positions of the linear axes. This cycle enables you to machine specific rotationally symmetrical contours in the active working plane. You can also run this cycle in the tilted working plane. The center of rotation is the starting point in the working plane at the time the cycle is called. Cycle 292 INTERPOLATION TURNING, CONTOUR FINISHING is run in milling mode and is CALL-active. After executing this cycle, the control deactivates the spindle coupling again.

Before using Cycle 292, you first need to define the desired contour in a subprogram and refer to this contour with Cycle 14 or SEL CONTOUR. Program the contour either with monotonically decreasing or monotonically increasing coordinates. Undercuts cannot be machined with this cycle. If you enter Q560=1, you can turn the contour and the cutting edge is oriented toward the circle center. If you enter Q560=0, you can mill the contour and the spindle is not oriented toward the circle center.



Cycle run, Q560=1: Contour turning



1 The control orients the tool spindle to the specified center of rotation. The specified angle Q336 is taken into account. If an "ORI" value is given in the turning tool table (toolturn.trn), it is also taken into account.
2 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
3 The control positions the tool at the contour start radius Q491, taking the selected machining operation (inside/outside Q529) and the set-up clearance to side Q357 into account. The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the control positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**
4 The control uses the interpolation turning cycle to machine the defined contour. In interpolation turning, the linear axes of the working plane move on a circle, whereas the spindle axis follows, it is oriented perpendicularly to the surface.
5 At the end point of the contour, the control retracts the tool perpendicularly to the set-up clearance.
6 Finally, the control retracts the tool to the clearance height.
7 The control automatically deactivates the coupling of the tool spindle to the linear axes.

Cycle run, Q560=0: Contour milling

1 The M3/M4 function programmed before the cycle call remains in effect.
2 No spindle stop and **no** oriented spindle stop will be performed. Q336 is not taken into account.
3 The control positions the tool at the contour start radius Q491, taking the selected machining operation (inside/outside Q529) and the set-up clearance to side Q357 into account. The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the control positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**
4 The control machines the defined contour using a rotating spindle (M3/M4). The principal axes of the working plane move on a circle, whereas the spindle axis does not follow.
5 At the end point of the contour, the control retracts the tool perpendicularly to the set-up clearance.
6 Finally, the control retracts the tool to the clearance height.

Please note while programming:

An example is provided at the end of this section, see Page 394.

Refer to your machine manual!

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

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

Software option 96 must be enabled.

With Q560=1, the control does not check whether the cycle is run with a rotating or stationary spindle. (Independent of **CfgGeoCycle** (no. 201000) - **displaySpindleError** (no. 201002))

Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.

The machine tool builder defines an M function for spindle orientation in the **CfgGeoCycle/mStrobeOrient** machine parameter (no. 201005).
If the value is >0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine tool builder). The control waits until the oriented spindle stop has been completed.
If you enter -1, the control will perform the oriented spindle stop.
If you enter 0, no action will be taken.
The control will under no circumstances output M5.

Please note that it is not possible to define programmed finishing allowances via the **FUNCTION TURNDATA CORR-TCS(WPL)** function. Program a finishing allowance for your contour directly in the cycle or by specifying a tool compensation (DXL, DZL, DRS) in the tool table.

When programming, remember to use only positive radius values.

Program the turning contour without tool radius compensation (RR/RL) and without APPR or DEP movements.

When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.

Program outside contours with a radius greater than 0.

Program inside contours with a radius greater than the tool radius.

Roughing operations with multiple passes are not possible in this cycle.

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds. Program Cycle 32 with HSC filter=1.

For inside contours, the control checks whether the active tool radius is less than half the diameter at the start of contour Q491 plus the set-up clearance on the side Q357. If the control determines that the tool is too large, the NC program will be canceled.

Please keep in mind that the axis angle must be equal to the tilt angle before calling the cycle. Only then is correct coupling of the axes possible.

If Cycle **8 MIRRORING** is active, the control does **not** perform the cycle for interpolation turning.

If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.

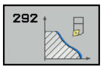
NOTICE

Danger of collision!

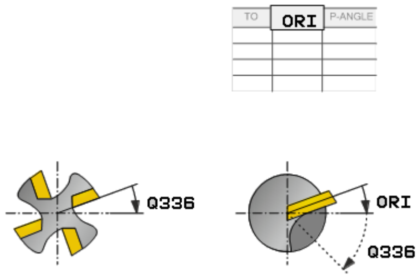
There is a risk of collision between tool and workpiece. The control does not automatically extend the described contour by a set-up clearance! At the beginning of the machining operation, the control positions the tool at rapid traverse FMAX to the contour starting point!

- Program an extension of the contour in the subprogram
- Make sure that there is no material at the contour starting point
- The center of the turning contour is the starting point in the working plane at the time the cycle is called

Cycle parameters



- **Q560 Spindle coupling (0=off, 1=on)?**: Specify whether the spindle should be coupled or not.
  0: Spindle coupling off (mill the contour)
  1: Spindle coupling on (turn the contour)
- **Q336 Angle for spindle orientation?**: The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool, and have defined the "ORI" value in the turning tool table (toolturn.trn), then it is taken into account for the oriented spindle stop. Input range 0.000 to 360.000
- **Q546 Reverse tool rotation direction?**: Direction of spindle rotation of the active tool:
  3: Clockwise rotation (M3)
  4: Anti-clockwise rotation (M4)
- **Q529 Machining operation (0/1)?**: Specify whether an inside or outside contour will be machined:
  +1: Inside machining
  0: Outside machining
- **Q221 Oversize for surface?**: Allowance in the working plane. Input range 0 to 99.9999
- **Q441 Infeed per revolution [mm/rev]?**: Dimension by which the control approaches the tool during one revolution. Input range: 0.001 to 99.999
- **Q449 Feed rate / cutting speed?** (mm/min): Feed rate relative to the contour starting point **Q491**. Input range: 0.1 to 99999.9. The feed rate of the tool's center path is adjusted according to the tool radius and **Q529 MACHINING OPERATION**. From these parameters, the control determines the programmed cutting speed at the diameter of the contour starting point.
  Q529=1: Feed rate of the tool's center path is reduced for inside machining
  Q529=0: Feed rate of the tool's center path is increased for outside machining



Example

63 CYCL DEF 292 CONTOUR.TURNG.INTRP.	
Q560=1	;SPINDLE COUPLING
Q336=0	;ANGLE OF SPINDLE
Q546=3	;CHANGE TOOL DIRECTN.
Q529=0	;MACHINING OPERATION
Q221=0	;SURFACE OVERSIZE
Q441=0.5	;INFEED
Q449=2000	;FEED RATE
Q491=0	;CONTOUR START RADIUS
Q357=2	;CLEARANCE TO SIDE
Q445=50	;CLEARANCE HEIGHT

- **Q491 Contour starting point (radius)?** (absolute value): Radius of the contour starting point (e.g. X-coordinate, if tool axis is Z). Input range: 0.9999 to 99999.9999
- **Q357 Safety clearance to the side?** (incremental): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth Input range 0 to 99999.9
- **Q445 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece; the tool retracts to this position at the end of the cycle. Input range -99999.9999 to 99999.9999

Machining variants

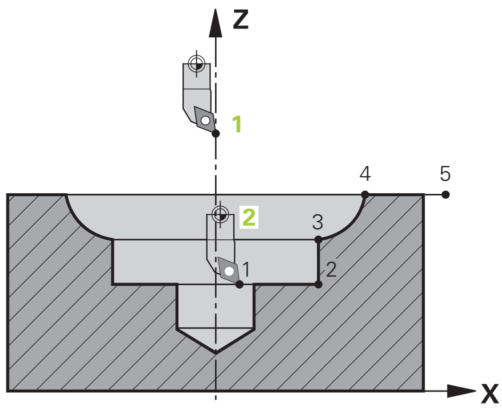
Before using Cycle 292, you first need to define the desired turning contour in a subprogram and refer to this contour with Cycle 14 or SEL CONTOUR. Describe the turning contour on the cross section of a rotationally symmetrical body. Depending on the tool axis, use the following coordinates to define the turning contour:

Tool axis used	Axial coordinate	Radial coordinate
Z	Z	X
X	X	Y
Y	Y	Z

Example: If you are using the tool axis Z, program the turning contour in the axial direction in Z and the radius of the contour in X.

You can use this cycle for inside and outside machining. The following information illustrates some of the notes given in the "Please note while programming" section. You can also find an example in "Example: Interpolation Turning Cycle 292", Page 394

Inside machining



- The center of rotation is the position of the tool in the working plane at the time the cycle is called **1**
- **After the cycle start, neither the indexable insert nor the spindle center must be moved into the center of rotation.** Keep this in mind while describing the contour **2**
- The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the control positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**

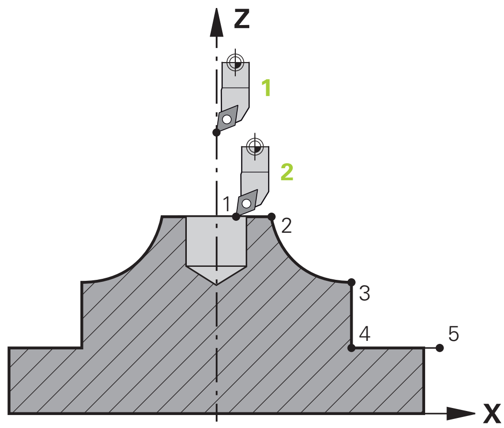
  When programming an inside contour, please also remember:
  - Program either monotonously increasing radial and axial coordinates, e.g. 1-5
  - Or program monotonously decreasing radial and axial coordinates, e.g. 5-1
  - Program inside contours with a radius greater than the tool radius.

Outside machining

- The center of rotation is the position of the tool in the working plane at the time the cycle is called **1**
- **After the cycle start, neither the indexable insert nor the spindle center must be moved into the center of rotation.** Keep this in mind while describing the contour **2**
- The described contour is not automatically extended by a set-up clearance. An extension of the contour must be programmed in the subprogram. At the beginning of the machining operation, the control positions the tool at rapid traverse in the tool axis direction to the contour starting point! **Make sure that there is no material at the contour starting point!**

  When programming an outside contour, please also remember:

  – Program monotonously increasing radial coordinates and monotonously decreasing axial coordinates, e.g. 1-5
  – Or program monotonously decreasing radial coordinates and monotonously increasing axial coordinates, e.g. 5-1
  – Program outside contours with a radius greater than 0.



Defining the tool

Overview

Depending on the entry for parameter Q560 you can either mill (Q560=0) or turn (Q560=1) the contour. For each of the two machining modes, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

Spindle coupling off, Q560=0

Milling: Define the milling cutter in the tool table as usual by entering the length, radius, toroid cutter radius, etc.

Spindle coupling on, Q560=1

Turning: The geometry data of the turning tool are converted to the data of a milling cutter. You now have the following three possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table a taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter Q336 of the cycle. For outside machining, the spindle orientation equals the value in Q336, and for inside machining the spindle orientation equals Q336+180.

  NOTICE

  Danger of collision!

  Collision may occur between the tool holder and workpiece with inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

  ▶ Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table a taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter Q336. For outside machining, the spindle orientation equals the value in Q336, and for inside machining the spindle orientation equals Q336+180.
- **Define a turning tool in the turning tool table (toolturn.trn)**

  If you are working with option 50, you can define the turning tool in the turning tool table (toolturn.trn). In this case, the spindle is oriented to the center of rotation by taking tool-specific data into account, such as the machining operation (TO in the turning tool table), the orientation angle (ORI in the turning tool table) and the parameter Q336.

  The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + Q336
Interpolation turning, inside	7	ORI + Q336 + 180
Interpolation turning, outside	7	ORI + Q336 + 180
Interpolation turning, inside	1	ORI + Q336
Interpolation turning, outside	8,9	ORI + Q336
Interpolation turning, inside	8,9	ORI + Q336

You can use the following tool types for interpolation turning:

- TYPE: ROUGH, with the machining directions TO: 1 or 7
- TYPE: FINISH, with the machining directions TO: 1 or 7
- TYPE: BUTTON, with the machining directions TO: 1 or 7

For inside contours, the control checks whether the active tool radius is less than half the diameter at the start of contour Q491 plus the set-up clearance on the side Q357. If the control determines that the tool is too large, the NC program will be canceled.

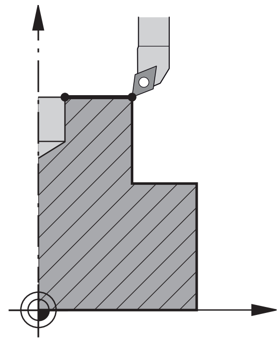
The following tool types cannot be used for interpolation turning: (error message "Function not possible with this tool type" is displayed)

- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD

12.7 COUPLING INTERPOLATION TURNING (Cycle 291, ISO: G291, software option 96)

Cycle run

Cycle **291 COUPLG.TURNG.INTERP.** couples the tool spindle to the position of the linear axes, or cancels this spindle coupling. With interpolation turning, the cutting edge is oriented to the center of a circle. The center of rotation is defined in the cycle by entering the coordinates Q216 and Q217. Cycle **291 COUPLG.TURNG.INTERP.** is run in milling mode and is CALL-active.



Cycle run if Q560=1:

1 The control first performs a spindle stop (M5).
2 The control orients the tool spindle to the specified center of rotation. The specified angle for spindle orientation Q336 is taken into account. If an "ORI" value is given in the tool table, it is also taken into account.
3 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4 To terminate the cycle, the coupling must be deactivated by the operator. (With Cycle 291, or end of program/internal stop.)

Cycle run if Q560=0:

1 The control deactivates the spindle coupling.
2 The tool spindle is no longer coupled to the position of the linear axes.
3 Machining with the interpolation turning cycle 291 is terminated.
4 If Q560=0, the parameters Q336, Q216, Q217 are irrelevant.

Please note while programming!

After defining Cycle 291 and **CYCL CALL** program the operation you wish to perform. To describe the circular motions of the linear axes, you can use linear/polar coordinates, for example. An example is provided at the end of this section, see Page 391.

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

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

Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.

Software option 96 must be enabled.

The machine tool builder defines an M function for spindle orientation in the **CfgGeoCycle/mStrobeOrient** machine parameter (no. 201005).
If the value is >0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine tool builder). The control waits until the oriented spindle stop has been completed.
If you enter -1, the control will perform the oriented spindle stop.
If you enter 0, no action will be taken.
The control will under no circumstances output M5.

Cycle 291 is CALL-active.

Programming of M3/M4 is not required. To describe the circular motions of the linear axes, you can use **CC** and **C** blocks, for example.

If you define the turning tool in the turning tool table (toolturn.trn), we recommend working with parameter Q561=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. With Q561=1 you can use radius compensation **RR** and **RL** when programming. (However, if you program Q561=0, then you cannot use radius compensation **RR** and **RL** when describing your contour. Additionally, you must program the movement of the tool center path **TCP** without spindle coupling. This kind of programming is much more complicated!)

If you programmed parameter Q561=1, you must program the following in order to conclude the interpolation turning machining operation:

- R0, cancels radius compensation
- Cycle 291 with parameters Q560=0 and Q561=0, deactivates spindle coupling
- **CYCL CALL**, for calling Cycle 291
- **TOOL CALL** overrides the conversion of parameter Q561

When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.

Program outside contours with a radius greater than 0.

Program inside contours with a radius greater than the tool radius.

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

Before cycle call, define a large tolerance with Cycle 32 for your machine to attain high contour speeds. Program Cycle 32 with HSC filter=1.

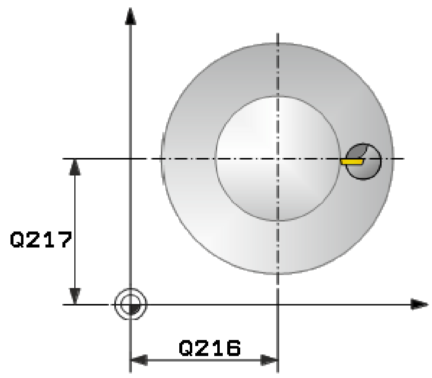
If Cycle **8 MIRRORING** is active, the control does **not** perform the cycle for interpolation turning.

If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.

Please keep in mind that the axis angle must be equal to the tilt angle before calling the cycle. Only then is correct coupling of the axes possible.

Cycle parameters

- **Q560 Spindle coupling (0=off, 1=on)?**: Specify whether the tool spindle is coupled to the position of the linear axes or not. If spindle coupling is active, the tool's cutting edge is oriented to the center of rotation.
  0: Spindle coupling off
  1: Spindle coupling on
- **Q336 Angle for spindle orientation?**: The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that a tooth is turned towards the center of rotation. If you work with a turning tool, and have defined the "ORI" value in the turning tool table (toolturn.trn), then it is taken into account for the oriented spindle stop. Input range 0.000 to 360.000
- **Q216 Center in 1st axis?** (absolute): Center of rotation in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q217 Center in 2nd axis?** (absolute): Center of rotation in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q561 Convert from turning tool** (0/1): Only relevant if you define the turning tool in the turning tool table (toolturn.trn). With this parameter you decide whether the value XL of the turning tool will be interpreted as radius R of a milling tool.
  0: No change; the turning tool is interpreted as it described in the turning tool table (toolturn.trn). In this case you may not use radius compensation **RR** or **RL**. Furthermore, you must describe the movement of the tool center path **TCP** without spindle coupling when programming. This kind of programming is much more difficult.
  1: The value XL from the turning tool table (toolturn.trn) will be interpreted as a radius R from a milling tool table. This makes it possible to use radius compensation **RR** and **RL** when programming your contour. This kind of programming is recommended.



Example

64 CYCL DEF 291 COUPLG.TURNG.INTERP.	
Q560=1	;SPINDLE COUPLING
Q336=0	;ANGLE OF SPINDLE
Q216=50	;CENTER IN 1ST AXIS
Q217=50	;CENTER IN 2ND AXIS
Q561=1	;TURNING TOOL CONVERSION

Defining the tool

Overview

Depending on the entry for parameter Q560 you can either activate (Q560=1) or deactivate (Q560=0) coupling for the interpolation turning cycle.

Spindle coupling off, Q560=0

The tool spindle is not coupled to the position of the linear axes.

> Q560=0: Deactivate the **COUPLING TURNING INTERPOLATION** cycle!

Spindle coupling on, Q560=1

A turning operation is executed with the tool spindle coupled to the position of the linear axes. If you set the parameter Q560 to 1, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table a taken into account (including delta values): length (L), radius (R), and corner radius (R2). The geometry data of the turning tool are converted to the data of a milling cutter. Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter Q336 of the cycle. For outside machining, the spindle orientation equals the value in Q336, and for inside machining the spindle orientation equals Q336+180.

  NOTICE

  Danger of collision!

  Collision may occur between the tool holder and workpiece with inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

  ▶ Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can perform interpolation turning with a milling tool. In this case the following data from the tool table a taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter Q336. For outside machining, the spindle orientation equals the value in Q336, and for inside machining the spindle orientation equals Q336+180.

- **Define a turning tool in the turning tool table (toolturn.trn)**

  If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case the orientation of the spindle to the center of turning takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table), parameter Q336, and parameter Q561.

If you define the turning tool in the turning tool table (toolturn.trn), we recommend working with parameter Q561=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. With Q561=1 you can use radius compensation **RR** and **RL** when programming. (However, if you program Q561=0, then you cannot use radius compensation **RR** and **RL** when describing your contour. Additionally, you must program the movement of the tool center path **TCP** without spindle coupling. This kind of programming is much more complicated!)

If you programmed parameter Q561=1, you must program the following in order to conclude the interpolation turning machining operation:

- R0, cancels radius compensation
- Cycle 291 with parameters Q560=0 and Q561=0, deactivates spindle coupling
- **CYCL CALL**, for calling Cycle 291
- **TOOL CALL** overrides the conversion of parameter Q561

If you programmed parameter Q561=1, you may only use the following types of tools:

- TYPE: ROUGH, FINISH, BUTTON with the machining directions TO: 1 or 8, XL>=0
- TYPE: ROUGH, FINISH, BUTTON with the machining directions TO: 7, XL<=0

The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + Q336
Interpolation turning, inside	7	ORI + Q336 + 180
Interpolation turning, outside	7	ORI + Q336 + 180
Interpolation turning, inside	1	ORI + Q336
Interpolation turning, outside	8	ORI + Q336
Interpolation turning, inside	8	ORI + Q336

You can use the following tool types for interpolation turning:

- TYPE: ROUGH, with the machining directions TO: 1, 7, 8
- TYPE: FINISH, with the machining directions TO: 1, 7, 8
- TYPE: BUTTON, with the machining directions TO: 1, 7, 8

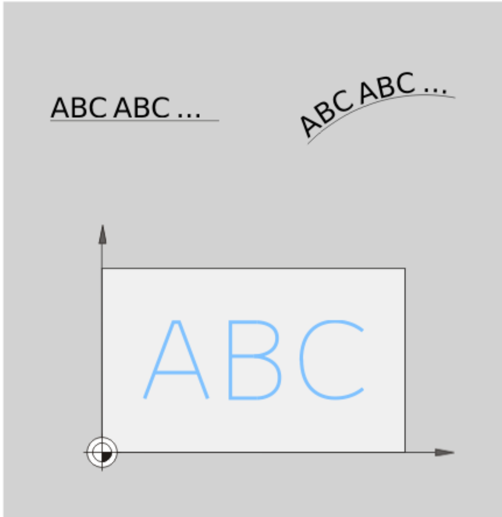
The following tool types cannot be used for interpolation turning: (error message "Function not possible with this tool type" is displayed)

- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD

12.8 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:

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

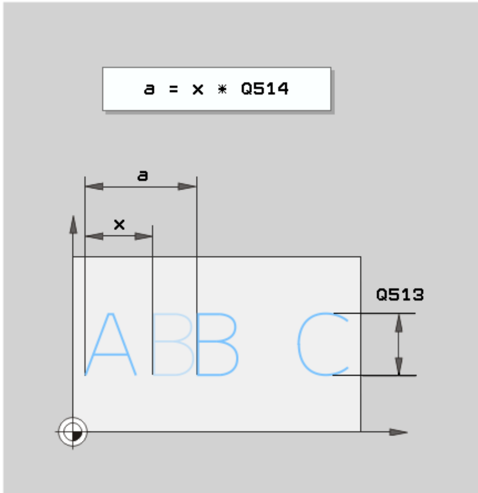
The text to be engraved can also be transferred with a string variable (**QS**).

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 362
- **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**



Example

62 CYCL DEF 225 ENGRAVING	
QS500="A"	;ENGRAVING TEXT
Q513=10	;CHARACTER HEIGHT
Q514=0	;SPACE FACTOR
Q515=0	;FONT
Q516=0	;TEXT ARRANGEMENT
Q374=0	;ANGLE OF ROTATION
Q517=0	;CIRCLE RADIUS
Q207=750	;FEED RATE FOR MILLNG
Q201=-0.5	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q200=2	;SET-UP CLEARANCE
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q367=+0	;TEXT POSITION
Q574=+0	;TEXT LENGTH

- **Q203 Workpiece surface coordinate?** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**
- **Q367 Reference for text position (0-6)?** Enter here the reference for the position of the text. Depending on whether the text will be engraved on an arc or a straight line (parameter Q516), the following values can be entered:
  If engraved on an arc, the text position refers to the following point:
  0 = Center of the circle
  1 = Bottom left
  2 = Bottom center
  3 = Bottom right
  4 = Top right
  5 = Top center
  6 = Top left
  If engraved on a straight line, the text position refers to the following point:
  0 = Bottom left
  1 = Bottom left
  2 = Bottom center
  3 = Bottom right
  4 = Top right
  5 = Top center
  6 = Top left
- **Q574 Maximum text length?** (mm/inch): Enter the maximum text length here. The control also takes into account parameter Q513 Character height. If Q513=0, the control engraves the text over exactly the length indicated in parameter Q574. The character height will be scaled accordingly. If Q513 is greater than zero, the control checks whether the actual text length exceeds the maximum text length entered in Q574. If that is the case, the control displays an error message.

Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers:

! # $ % & ‘ ( ) * + , - . / : ; < = > ? @ [\ ] _ ß CE

> 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 counter reading

You can engrave the current counter reading found in the MOD menu with Cycle 225.

For this purpose program Cycle 225 as usual and enter e.g. the following for the text to be engraved: **%count2**

The number after **%count** indicates how many digits the control will engrave. The maximum is of nine digits.

Example: If you program **%count9** in the cycle with a momentary counter reading of 3, the control will engrave the following: 000000003

> In the Test Run operating mode, the control only simulates the counter reading you specified directly in the NC program. The counter reading from the MOD menu is not taken into account.
>
> In the SINGLE BLOCK and FULL SEQ. and Program Run, Single Block operating modes, the control will take the counter reading from the MOD menu into account.

12.9 FACE MILLING (Cycle 232, ISO: G232)

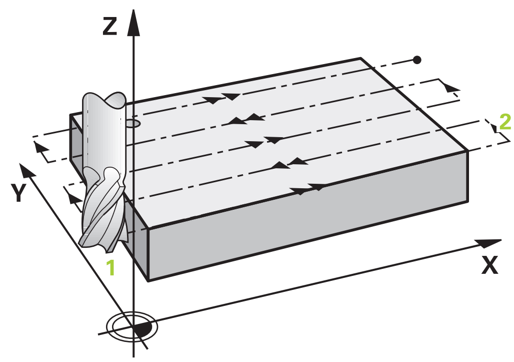
Cycle run

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

- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** using positioning logic: If the current position in the spindle axis is further away from the workpiece than the 2nd set-up clearance, the control positions the tool first in the machining plane and then in the spindle axis. Otherwise, it first moves it to the 2nd set-up clearance and then in the machining plane. The starting point in the machining plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
2 The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

Strategy Q389=0



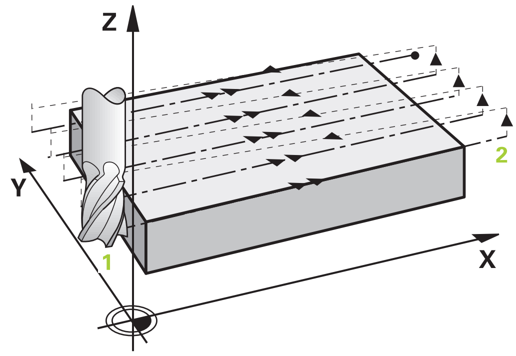
3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategy Q389=1

3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **at the edge** of the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.
4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**. The motion to the next pass again occurs at the edge of the workpiece.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategy Q389=2

3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves it at the pre-positioning feed rate directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then returns to the current infeed depth and moves in the direction of end point **2**
6 The process is repeated until the programmed surface has been machined completely. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.



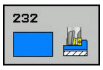
Please note while programming:

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

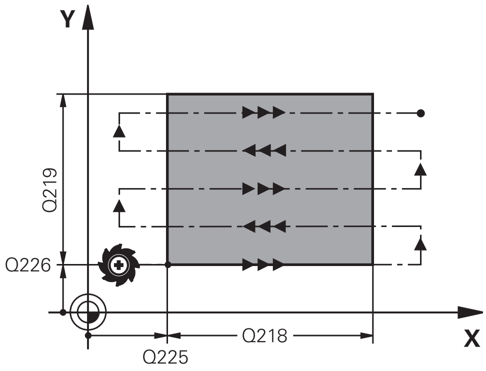
If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).

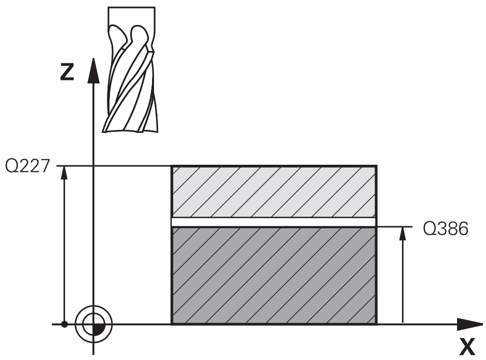
Program Q227 greater than Q386. The control will otherwise display an error message.

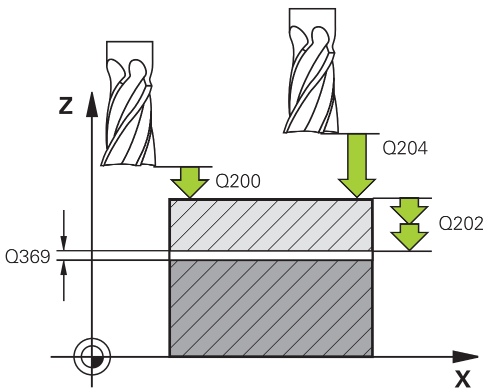
Cycle parameters



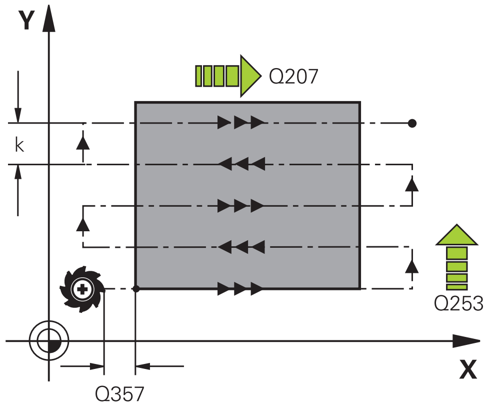
- **Q389 Machining strategy (0/1/2)?**: Determine how the control should machine the surface:
  0: Meander machining, stepover at the positioning feed rate outside the surface being machined
  1: Meander machining, stepover at the feed rate for milling at the edge of the surface being machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate
- **Q225 Starting point in 1st axis?** (absolute): Starting point coordinate of the surface to be machined in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q226 Starting point in 2nd axis?** (absolute): Starting point coordinate of the surface to be machined in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q227 Starting point in 3rd axis?** (absolute): Coordinate of the workpiece surface used to calculate the infeeds. Input range: -99999.9999 to 99999.9999
- **Q386 End point in 3rd axis?** (absolute): Coordinate in the spindle axis to which the surface is to be face-milled. Input range: -99999.9999 to 99999.9999
- **Q218 First side length?** (incremental): Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path referencing the **starting point in the 1st axis**. Input range: -99999.9999 to 99999.9999
- **Q219 Second side length?** (incremental): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **STARTNG PNT 2ND AXIS**. Input range: -99999.9999 to 99999.9999
- **Q202 Maximum plunging depth?** (incremental): **Maximum** infeed per cut. The control calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. Input range: 0 to 99999.9999
- **Q369 Finishing allowance for floor?** (incremental): Distance used for the last infeed. Input range: 0 to 99999.9999







- **Q370 Max. path overlap factor?**: **Maximum** stepover factor k. The control calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. cutter radius when using a face-milling cutter), the control reduces the stepover accordingly. Input range: 0.1 to 1.9999
- **Q207 Feed rate for milling?**: Traversing speed of the tool in mm/min during milling. Input range 0 to 99999.999 alternatively **FAUTO**, **FU**, **FZ**
- **Q385 Finishing feed rate?**: Traversing speed of the tool in mm/min while milling the last infeed. Input range: 0 to 99999,9999; alternatively **FAUTO**, **FU**, **FZ**
- **Q253 Feed rate for pre-positioning?**: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (Q389=1), the control uses the cross feed rate for milling Q207. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- **Q200 Set-up clearance?** (incremental): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy Q389=2, the control moves the tool to the set-up clearance above the current plunging depth to the starting point of the next pass. Input range: 0 to 99999.9999
- **Q357 Safety clearance to the side?** (incremental) Parameter Q357 affects the following situations:
  Approaching the first plunging depth: Q357 is the lateral distance between tool and workpiece
  Roughing with the Q389=0-3 roughing strategies: The surface to be machined is extended in **Q350** MILLING DIRECTION by the value from Q357 if no limit has been set in that direction
  Side finishing: The paths will be extended by Q357 in **Q350** MILLING DIRECTION.
  Input range: 0 to 99999.9999
- **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range: 0 to 99999.9999; alternatively **PREDEF**



Example

71 CYCL DEF 232 FACE MILLING	
Q389=2	;STRATEGY
Q225=+10	;STARTNG PNT 1ST AXIS
Q226=+12	;STARTNG PNT 2ND AXIS
Q227=+2.5	;STARTNG PNT 3RD AXIS
Q386=-3	;END POINT 3RD AXIS
Q218=150	;FIRST SIDE LENGTH
Q219=75	;2ND SIDE LENGTH
Q202=2	;MAX. PLUNGING DEPTH
Q369=0.5	;ALLOWANCE FOR FLOOR
Q370=1	;MAX. OVERLAP
Q207=500	;FEED RATE FOR MILLNG
Q385=800	;FINISHING FEED RATE
Q253=2000	;F PRE-POSITIONING
Q200=2	;SET-UP CLEARANCE
Q357=2	;CLEARANCE TO SIDE
Q204=2	;2ND SET-UP CLEARANCE

12.10 ASCERTAIN THE LOAD (Cycle 239, ISO: G239, software option 143)

Cycle run

The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of the table axes. With option # 143 LAC (Load Adaptive Control) and Cycle 239 ASCERTAIN THE LOAD, the control is able to automatically determine and adjust the actual mass inertia of the load, the actual friction forces and the maximum axis acceleration, or reset the feedforward and controller parameters. In this way, you can optimally react to major load changes. The control performs a weighing procedure to ascertain the weight acting on the axes. With this weighing run, the axes return a certain path. Your machine manufacturer defines the specific movements. Before weighing, the axes are moved to a position, if required, where there is no danger of collision during the weighing procedure. This safe position is defined by the machine tool builder.

In addition to adjusting the control parameters, with LAC the maximum acceleration is also adjusted in accordance with the weight. This enables the dynamics to be accordingly increased with low load to increase productivity.

Parameter Q570 = 0

1. There is no physical movement of the axes.
2. The control resets the LAC.
3. The TNC activates feedforward parameters and, if applicable, controller parameters that ensure a safe movement of the axes concerned, regardless of the load condition—the parameters set with Q570=0 are **independent** of the current load.
4. These parameters can be useful during the setup procedure or after the completion of an NC program.

Parameter Q570 = 1

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

Please note while programming:

The machine must have been prepared by the machine tool builder for this cycle.

Cycle 239 works only together with Option 143 LAC (Load Adaptive Control).

NOTICE

Danger of collision!

The cycle can execute extensive movements in several axes at rapid traverse!

- Ask your machine manufacturer about the type and extent of Cycle 239 before using it
- Before the cycle starts, the control moves to a safe position, if applicable. The machine tool builder determines this position.
- Set the potentiometers for feed-rate and rapid-traverse override to at least 50 % to ensure a correct ascertainment of the load.

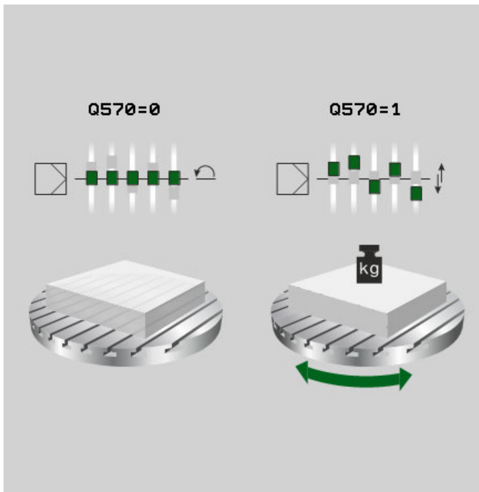
Cycle 239 becomes effective immediately after its definition.

If you are using the mid-program startup function and the control thus skips Cycle 239 in the block scan, the control will ignore this cycle—no weighing procedure will be performed.

Cycle 239 supports the determination of the load on synchronized axes (gantry axes) if they have only one common position encoder (torque master slave).

Cycle parameters

- **Q570 Load (0 = Delete/1 = Ascertain)?**: Specify whether the control should perform an LAC (Load adaptive control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters should be reset:
  0: Reset LAC; the values most recently ascertained by the control are reset, and the control uses load-independent feedforward and controller parameters
  1: Perform a weighing run; the control moves the axes and thus ascertains the feedforward and controller parameters depending on the current load. The values ascertained are activated immediately.



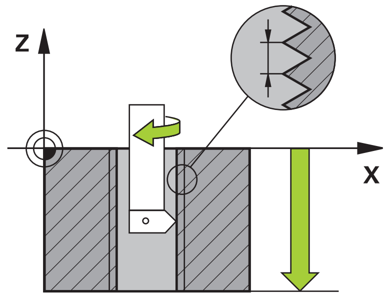
Example

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

12.11 THREAD CUTTING (Cycle 18, ISO: G18)

Cycle run

Cycle **18** THREAD CUTTING moves the tool with servo-controlled spindle from the momentary position with active speed to the specified depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.



Please note while programming:

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

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

The spindle speed potentiometer is inactive.

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

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

NOTICE

Danger of collision!

A collision may occur if you do not program pre-positioning before calling Cycle 18. Cycle 18 does not perform approach and departure motion.

- Pre-position the tool before the start of the cycle.
- The tool moves from the current position to the entered depth after the cycle is called

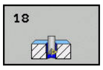
NOTICE

Danger of collision!

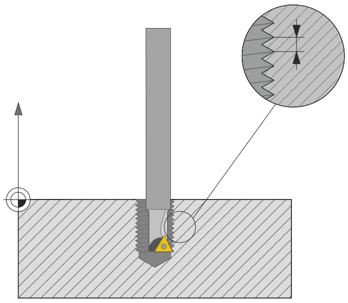
If the spindle was switched on before calling the cycle, Cycle 18 switches the spindle off and the cycle works with a stationary spindle! Cycle 18 switches the spindle on again at the end if it was switched on before cycle start.

- Before starting this cycle, be sure to program a spindle stop! (For example with M5.)
- At the end of Cycle 18, the control restores the spindle to its state at cycle start. If the spindle was switched off before this cycle, the control will switch it off again at the end of Cycle 18.

Cycle parameters



- Boring depth (incremental): Enter the thread depth based on the current position. Input range: -99999 ... +99999
- Thread pitch: Enter the pitch of the thread. The algebraic sign entered here differentiates between right-hand and left-hand threads:
  + = right-hand thread (M3 with negative hole depth)
  - = left-hand thread (M4 with negative hole depth)



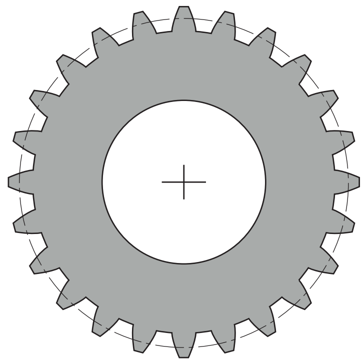
Example

25 CYCL DEF 18.0 THREAD CUTTING
26 CYCL DEF 18.1 DEPTH = -20
27 CYCL DEF 18.2 PITCH = +1

12.12 FUNDAMENTALS ON MACHINING GEARS (software option 157)

Fundamentals

Refer to your machine manual.
This function must be enabled and adapted by the machine tool builder.



For the cycles, option 157 Gear Cutting is required. If you would like to use these cycles in turning mode, you also need option 50. In milling mode, the tool spindle is the master spindle, in turning mode, it is the workpiece spindle. The other spindle is called slave spindle. Depending on the operating mode, you program the speed or the cutting speed with a **TOOL CALL S** or **FUNCTION TURNDATA SPIN**.

To orient the I-CS coordinate system, Cycles 286 and 287 use the precession angle that is also affected by Cycles 800 and 801 in turning mode. At the end of the cycle, the control resets the precession angle to its state at the beginning of the cycle. If one of these cycles is aborted, the precession angle will also be reset.

The axis crossing angle is the angle between workpiece and tool. It results from the angle of inclination of the tool and the angle of inclination of the gear. Based on the required axis crossing angle, Cycles 286 and 287 calculate the required inclination of the rotary axis at the machine. The cycles will always position the first rotary axis starting from the tool.

The gear itself will first be described in Cycle 285 **DEFINE GEAR**. Then, program either Cycle 286 **GEAR HOBBING** or 287 **GEAR SKIVING**.

Before calling a cycle, be sure to program the following:

- Call a tool with **TOOL CALL**
- Select turning or milling mode with **FUNCTION MODE TURN / MILL**
- In turning mode: select the speed/cutting speed with **FUNCTION TURNDATA SPIN** or in milling mode: **TOOL CALL S**
- Spindle direction of rotation, e.g. **M3** or **M303**
- If required, call the **CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM** cycle**RESET ROTARY COORDINATE SYSTEM**
- Perform pre-positioning for the cycle depending on your selection of **MILL** or **TURN**
- Call the **CYCL DEF 285 DEFINE GEAR** cycle
- Call the **CYCL DEF 286 GEAR HOBBING** or **CYCL DEF 287 GEAR SKIVING** cycle

Please note while programming!

Before calling the cycle, set the preset to the center of rotation of the workpiece spindle.

Please note that the slave spindle will continue to rotate after the end of the cycle. If you want to stop the spindle before the end of the program, make sure to program a corresponding M function.

All machining feed rates are given in mm/rev. of the tool spindle.

The cycles automatically define the direction and path for a **LiftOff**. This function must be enabled by the machine tool builder. In addition, **LiftOff** must be permitted for the respective tool.

NOTICE

Danger of collision!

If you do not pre-position the tool to a safe position, a collision between tool and workpiece (fixtures) may occur during tilting.

- Pre-position the tool to a safe position

NOTICE

Danger of collision!

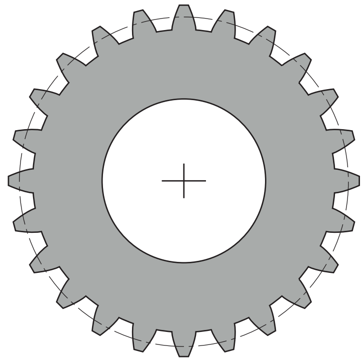
If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance Q200!

- Make sure to clamp the workpiece in such a way that it projects far enough from the fixture and no collision can occur between tool and fixture.

12.13 DEFINE GEAR (Cycle 285, ISO: G285, software option 157)

Cycle run

With Cycle 285 **DEFINE GEAR** you describe the geometry of the gearing system. The tool will be described in Cycle 286 **GEAR HOBBING** or in Cycle 287 **GEAR SKIVING** and in the tool table (TOOL.T).



Please note while programming!

You must specify values for module and number of teeth. If the outside diameter (diameter of the addendum circle) and the tooth height are defined as 0, normal running gears (DIN 3960) will be machined. If you want to machine gearing systems that differ from this standard, you can define the corresponding geometry by specifying the outside diameter **Q542** and the tooth height **Q563**.

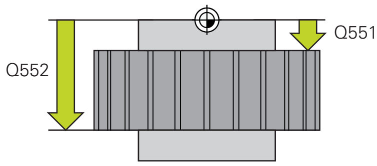
Define the tool as a milling cutter in the tool table.

If the algebraic signs of the two input parameters **Q541** and **Q542** are contradictory, the cycle will be aborted with an error message.

This cycle is DEF-active. The values of these Q parameters will only be read when a CALL-active machining cycle is executed. If you overwrite these input parameters after the cycle definition and before calling the machining cycle, the gear geometry will be modified.

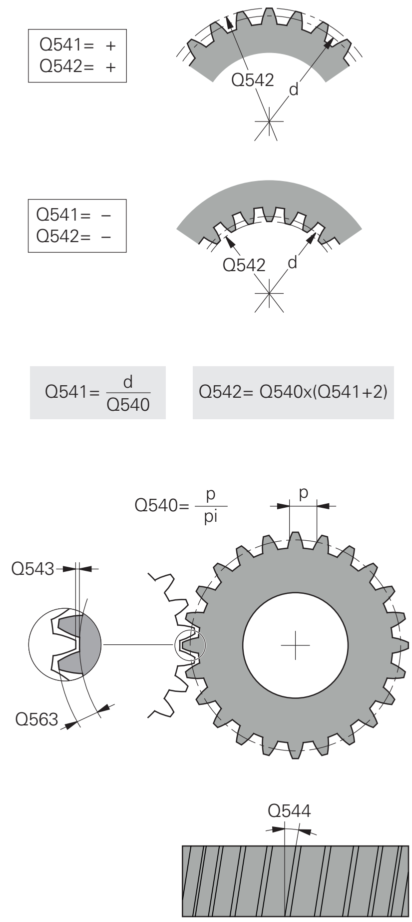
The two cycle parameters, **Q541 NUMBER OF TEETH** and **Q542 OUTSIDE DIAMETER**, must have the same algebraic sign. Otherwise, an error message will be issued.

Cycle parameters



- **Q551 Starting point in Z?**: Starting point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- **Q552 End point in Z?**: End point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- **Q540 Module?**: Define the gear: Module of the gear wheel. Input range 0 to 99.9999
- **Q541 Number of teeth?**: Number of teeth. This parameter depends on **Q542**.
  +: If the number of teeth is positive and parameter **Q542** is also positive, you define an outer gear
  -: If the number of teeth is negative and parameter **Q542** is negative as well, you define an inner gear
  Input range: -9999,9999 to +9999,9999
- **Q542 Outside diameter?**: Diameter of the tip circle of the gear wheel. This parameter depends on **Q541**.
  +: If the outside diameter is positive and parameter **Q541** is also positive, you define an outer gear
  -: If the outside diameter is negative and parameter **Q541** is negative as well, you define an inner gear
  Input range: -9999,9999 to +9999,9999

- **Q563 Tooth height?** Distance from the bottom edge of the tooth to the top edge. Input range: 0 to 999.9999
- **Q543 Trough-to-tip clearance?**: Define the gear: Distance between the tip circle of the gear to be cut and the root circle of the mating gear. Input range 0 to 9.9999
- **Q544 Angle of inclination?**: Define the gear: Angle by which teeth of a helical gear helical teeth are inclined relative to the direction of the axis (For straight-cut gears this angle is 0°.) Input range: -60 to +60



Example

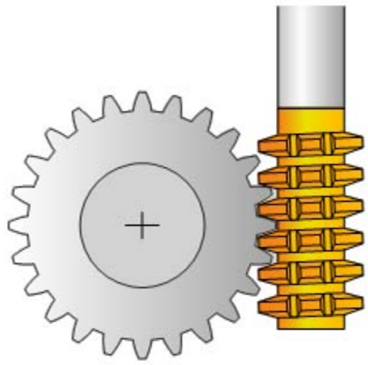
```
63 CYCL DEF 285  ZAHNRAD
    DEFINIEREN
    Q551=0       ;STARTING POINT IN Z
    Q552=-10     ;END POINT IN Z
    Q540=1       ;MODULE
    Q541=+10     ;NUMBER OF TEETH
    Q542=0       ;OUTSIDE DIAMETER
    Q563=0       ;TOOTH HEIGHT
    Q543=+0.17;TROUGH-TIP CLEARANCE
    Q544=0       ;ANGLE OF INCLINATION
```

12.14 GEAR HOBBING (Cycle 286, ISO: G286, software option 157)

Cycle run

With Cycle 286 **GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. You can select the machining strategy and the machining side in the cycle.
The machining process for gear hobbing is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction.

Cycle run:



1 The control positions the tool in the tool axis to clearance height **Q260** at the feed rate **FMAX**. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the **FMAX** feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control then tilts the working plane at the feed rate **Q253**.
4 The control positions the tool at the feed rate **FMAX** to the starting point in the working plane.
5 The control then moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**.
6 The control now moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552**+**Q200** (**Q551** and **Q552** are defined in Cycle 285).

   Further information: "DEFINE GEAR (Cycle 285, ISO: G285, software option 157)", Page 376
7 When the tool reaches the end point, it is retracted at the feed rate **Q253** and returns to the starting point.
8 The control repeats the steps 5 to 7 until the defined gear is completed.
9 Finally, the control retracts the tool to the clearance height **Q260** at the feed rate **FMAX**.
10 When you machine helical gears, the tilts of the rotary axes will not be reset at the end of the cycle.
11 You need to move the tool to a safe height and reset the tilting of the working plane.

Please note while programming!

Cycle 286 can be used both in milling and turning mode. The cycle is CALL-active.

In order to ensure constant engagement of the cutting edge of a tool, you need to define a very small path in cycle parameter **Q554 SYNCHRONOUS SHIFT**.

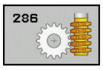
In turning mode, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM** before calling Cycle 286.

Avoid master spindle speeds of less than 6 rpm in turning mode. Otherwise, it is not possible to reliably use a feed rate in mm/rev. If you need to use a lower master spindle speed, rather use the cycle in milling mode than in turning mode.

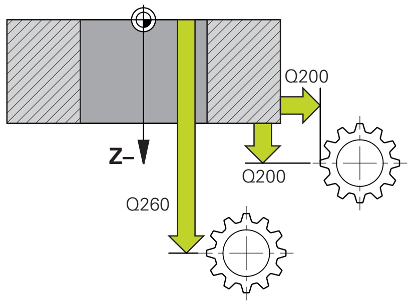
Make sure to program the direction of rotation of the master spindle before the cycle start.

If you program **FUNCTION TURNDATA SPIN VCONST:OFF S15**, the spindle speed of the tool is calculated as **Q541** x S. With **Q541**=238 and S=15, this would result in a tool spindle speed of 3570 rpm.

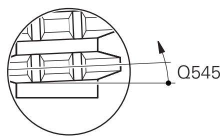
Cycle parameters



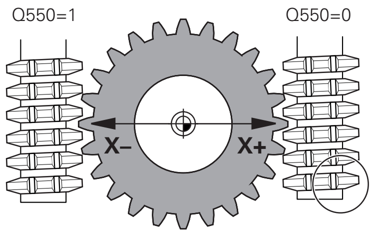
- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize



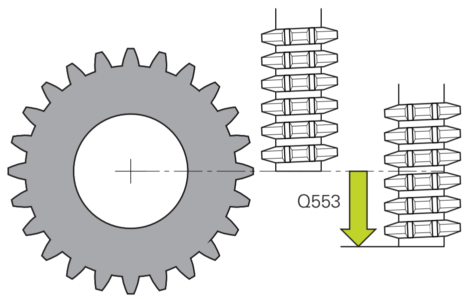
- **Q200 Set-up clearance?** (incremental): Distance for retraction and pre-positioning Input range 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). Input range: -99999.9999 to 99999.9999



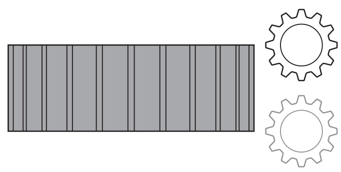
- **Q545 Tool lead angle?**: Define the tool: Angle of the tooth sides of the gear hob. Enter this value in decimal notation. (Example: 0°47'=0.7833) Input range: -60.0000 to +60.0000



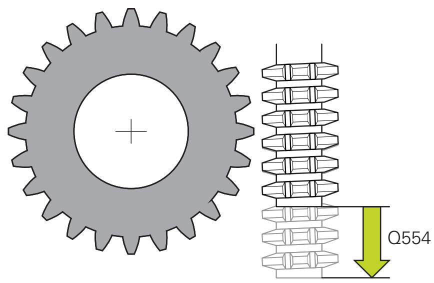
- **Q546 Reverse spindle rotation dir.?**: Reverse the direction of rotation of the slave spindle:
  0: Direction of rotation is not reversed
  1: Direction of rotation is reversed
  Further information: "Verifying and changing directions of rotation of the spindles", Page 384
- **Q547 Angle offset of tool spindle?**: Angle by which the control rotates the workpiece at the beginning of cycle. Input range -180.0000 to +180.0000



- **Q550 Machining side (0=pos./1=neg.)?**: Define the side on which the machining operation is to be performed.
  0: Positive machining side of the reference axis in the I-CS
  1: Negative machining side of the reference axis in the I-CS



- **Q533 Preferred dir. of incid. angle?**: Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on the machine. In general, there are always two possible solutions. Using parameter Q533, configure which of the solution options the control should use:
  0: Option with the shortest distance from the current position
  -1: Option in the range between 0° and -179.9999°
  +1: Option in the range between 0° and +180°
  -2: Option in the range between -90° and -179.9999°
  +2: Option in the range between +90° and +180°



- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN)
- **Q253Feed rate for pre-positioning?**: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Input in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q553 TOOL:L offset, machining start?** (incremental): Define the length offset (L OFFSET) from which on the tool will machine. The tool will be offset in the longitudinal direction by this value. Input range: 0 to 999.9999
- **Q554 Path for synchronous shift?**: Define by which distance the gear hob will be offset in its axial direction during machining. This way, tool wear can be distributed over this area of the cutting edges. For helical gears, it is thus possible to limit the cutting edges used for machining. Entering 0 deactivates the synchronous shift function.
  Input range: -99.9999 to +99.9999
- **Q548 Tool shift for roughing?**: Specify the number of cutting edges by which the control will shift the roughing tool in its axial direction. The shift will be performed incrementally, referencing parameter **Q553**. Entering 0 deactivates the shift function. Input range: -99 to +99
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q488 Feed rate for plunging**: Feed rate for tool infeed. The control interprets the feed rate in mm per revolution. Input range 0 to 99999.999 alternatively **FAUTO**, **PREDEF**
- **Q478 Roughing feed rate?**: Feed rate during roughing. The control interprets the feed rate in mm per revolution. Input range: 0 to 99999.999; alternatively **FAUTO**, **PREDEF**
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999

Example

63 CYCL DEF 286 GEAR SKIVING	
Q215=0	;MACHINING OPERATION
Q200=+2	;SET-UP CLEARANCE
Q260=+100	;CLEARANCE HEIGHT
Q545=0	;TOOL LEAD ANGLE
Q546=0	;CHANGE ROTATION DIR.
Q547=0	;ANG. OFFSET, SPINDLE
Q550=1	;MACHINING SIDE
Q533=0	;PREFERRED DIRECTION
Q530=2	;INCLINED MACHINING
Q253=750	;F PRE-POSITIONING
Q553=10	;TOOL LENGTH OFFSET
Q554=0	;SYNCHRONOUS SHIFT
Q548=0	;ROUGHING SHIFT
Q463=1	;MAX. CUTTING DEPTH
Q488=0.3	;PLUNGING FEED RATE
Q478=0.3	;ROUGHING FEED RATE
Q483=0.4	;OVERSIZE FOR DIAMETER
Q505=0.2	;FINISHING FEED RATE
Q549=0	;FINISHING SHIFT

- **Q505 Finishing feed rate?**: Feed rate during finishing. The control interprets the feed rate in mm per revolution. Input range: 0 to 99999.999; alternatively **FAUTO**, **PREDEF**
- **Q549 Tool shift for finishing?**: Specify the number of cutting edges by which the control will shift the finishing tool in its longitudinal direction. The shift will be performed incrementally, referencing parameter **Q553**. Entering 0 deactivates the shift function. Input range: -99 to +99

Verifying and changing directions of rotation of the spindles

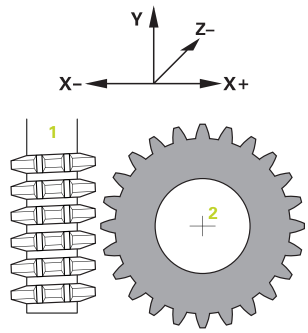
Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

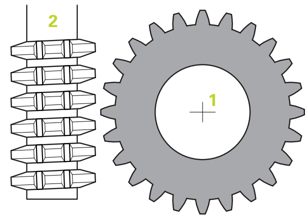
Changing the direction of rotation in milling mode:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation. Changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle.
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Changing the direction of rotation in turning mode:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation. Changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle.
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.





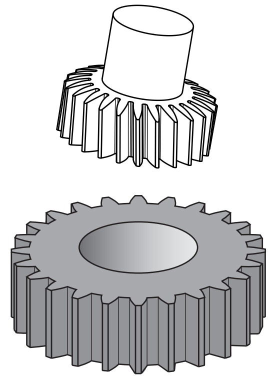
> If required, define a low spindle speed to make sure that the direction of rotation is correct.

12.15 GEAR SKIVING (Cycle 287, ISO: G287, software option 157)

Cycle run

With Cycle 287 **GEAR SKIVING**, you can machine cylindrical gears or helical gears with any angles. Chips are formed both by the axial feed of the tool and by the skiving movement.

You can select the machining side in the cycle. The machining process for gear skiving is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction.



Cycle run:

1 The control positions the tool in the tool axis to clearance height **Q260** at the feed rate **FMAX**. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the **FMAX** feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control then tilts the working plane at the feed rate **Q253**.
4 The control positions the tool at the feed rate **FMAX** to the starting point in the working plane.
5 The control then moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**.
6 The control approaches the approach path. This path is calculated by the control. The approach length is the distance from the initial scratch to the full plunging depth.
7 The control moves the tool at the defined feed rate to gear-skive the workpiece in longitudinal direction. For the infeed for the first cut **Q586**, the control moves the tool at the first feed rate **Q588**. In addition, the control will calculate intermediate values for the next cuts (both infeed and feed rate). The control performs these calculations automatically. The intermediate feed rate values, however, depend on the factor for feed-rate adaptation **Q580**. When the tool has reached the last infeed **Q587**, feed rate **Q589** will be used for the last cut.
8 The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552** (**Q551** and **Q552** are defined in Cycle 285). In addition to the starting point, the control will consider the approach length. The purpose is to avoid that the tool plunges down to the machined diameter into the workpiece. The control calculates this path automatically.

9 At the end of machining, the tool approaches the idle travel path. The idle travel distance serves to completely machine the gearing system to the end point. The control calculates this path automatically, too.
10 When the tool reaches the end point, it is retracted at the feed rate **Q253** and returns to the starting point.
11 Finally, the control retracts the tool to the clearance height **Q260** at the feed rate FMAX.
12 When you machine helical gears, the tilts of the rotary axes will not be reset at the end of the cycle.
13 You need to move the tool to a safe height and reset the tilting of the working plane.

Please note while programming!

Cycle 287 can be used both in milling and turning mode. The cycle is CALL-active.

In turning mode, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM** before calling Cycle 287.

Make sure to program the direction of rotation of the master spindle before the cycle start.

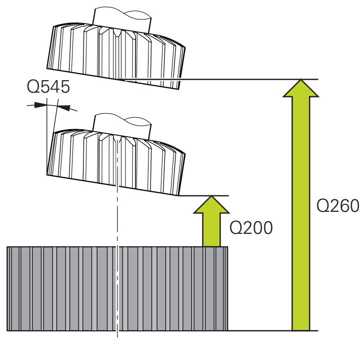
The larger the factor in **Q580** FEED-RATE ADAPTION, the earlier the control will adapt the feed rate to the feed rate for the last cut. The recommended value is 0.2.

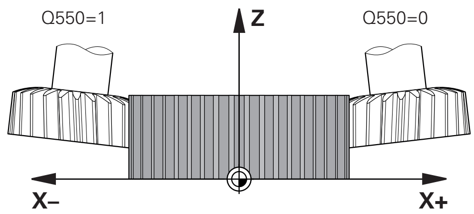
When defining the tool, make sure to specify the number of cutting edges as indicated in the tool table.

The speed ratio between tool and workpiece results from the number of teeth of the gear wheel and the number of cutting edges of the tool.

Cycle parameters

- **Q240 Number of cuts?** Number of cuts until the end depth is reached
  0: The control automatically calculates the minimum number of cuts
  1: One cut
  2: Two cuts, the control will only consider the infeed for the first cut **Q586**. In this case, the infeed for the last cut **Q587** will not be taken into account
  3-99999: Programmed number of cuts
- **Q584 Number of the first cut?**: Specify the number of the cut to be machined first. Input range: 1 to 999
- **Q585 Number of the last cut?**: Specify the number of the cut to be machined last. Input range: 1 to 999
- **Q200 Set-up clearance?** (incremental): Distance for retraction and pre-positioning 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
- **Q545 Tool lead angle?**: Define the tool: Angle of the tooth sides of the gear skiving hob. Enter this value in decimal notation. (Example: 0°47'=0.7833) Input range: -60.0000 to +60.0000
- **Q546 Reverse spindle rotation dir.?**: Reverse the direction of rotation of the slave spindle:
  0: Direction of rotation is not reversed
  1: Direction of rotation is reversed
  Further information: "Verifying and changing directions of rotation of the spindles", Page 390
- **Q547 Angle offset of tool spindle?**: Angle by which the control rotates the workpiece at the beginning of cycle. Input range -180.0000 to +180.0000
- **Q550 Machining side (0=pos./1=neg.)?**: Define the side on which the machining operation is to be performed.
  0: Positive machining side of the reference axis in the I-CS
  1: Negative machining side of the reference axis in the I-CS





Example

```
63 CYCL DEF 287 ZAHNRAD
   WALZSCHAELEN
   Q240=0      ;NUMBER OF CUTS
   Q584=+1     ;NO. OF FIRST CUT
   Q585=+999   ;NO. OF LAST CUT
   Q200=2      ;SET-UP CLEARANCE
   Q260=+100   ;CLEARANCE HEIGHT
   Q545=0      ;TOOL LEAD ANGLE
   Q546=0      ;CHANGE ROTATION DIR.
   Q547=0      ;ANG. OFFSET, SPINDLE
   Q550=+1     ;MACHINING SIDE
   Q533=0      ;PREFERRED DIRECTION
   Q530=+2     ;INCLINED MACHINING
   Q253=+750   ;F PRE-POSITIONING
   Q586=+1     ;FIRST INFEED
   Q587=+0.1   ;LAST INFEED
   Q588=+0.2   ;FIRST FEED RATE
   Q589=+0.05;LAST FEED RATE
   Q580=+0.2   ;FEED-RATE ADAPTION
```

- **Q533 Preferred dir. of incid. angle?**: Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on the machine. In general, there are always two possible solutions. Using parameter Q533, configure which of the solution options the control should use:
  0: Option with the shortest distance from the current position
  -1: Option in the range between 0° and -179.9999°
  +1: Option in the range between 0° and +180°
  -2: Option in the range between -90° and -179.9999°
  +2: Option in the range between +90° and +180°
- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN)
- **Q253Feed rate for pre-positioning?**: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Input in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q586 Infeed for first cut?** (incremental): Infeed value for the first cut. Input range: 0.001 to 99.999
- **Q587 Infeed for last cut?** (incremental): Infeed value for the last cut. Input range: 0.001 to 99.999
- **Q588 Feed rate for first cut?**: Feed rate for the first cut. The control interprets the feed rate in mm per revolution. Input range: 0.001 to 99.999

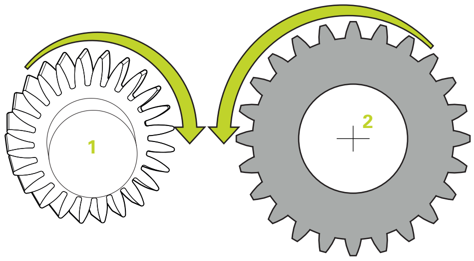
- **Q589 Feed rate for last cut?**: Feed rate for the last cut. The control interprets the feed rate in mm per revolution. Input range: 0.001 to 99.999
- **Q580 Factor for feed-rate adaptation?**: Since it is necessary to reduce the feed rate with increasing cut numbers, you can use this factor to define a feed rate reduction. The larger the value, the earlier the control will adapt the feed rate to the feed rate for the last feed rate. Input range: 0.000 to 1.000

Verifying and changing directions of rotation of the spindles

Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

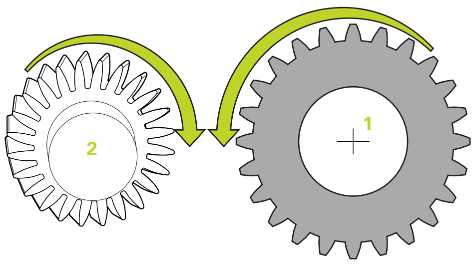
Changing the direction of rotation in milling mode:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation. Changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle.
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter Q546.



Changing the direction of rotation in turning mode:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation. Changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle.
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter Q546.



> If required, define a low spindle speed to make sure that the direction of rotation is correct.

12.16 Programming examples

Example: Interpolation turning with Cycle 291

The following NC program illustrates the use of Cycle **291 COUPLG.TURNG.INTERP.** This programming example illustrates the machining of an axial recess and a radial recess.

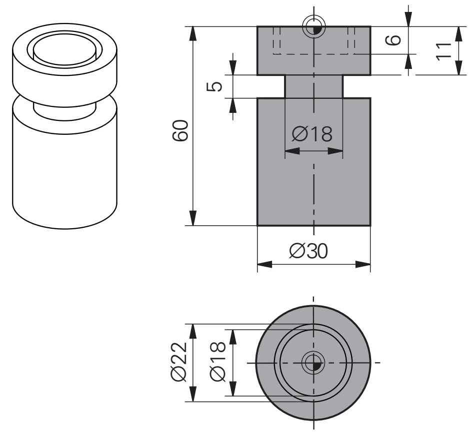
Tools

- Turning tool as defined in toolturn.trn: Tool no. 10: TO:1, ORI:0, TYPE:ROUGH, tool for axial recesses
- Turning tool as defined in toolturn.trn: Tool no. 11: TO:8, ORI:0, TYPE:ROUGH, tool for radial recesses

Program run

- Tool call: tool for axial recess
- Start of interpolation turning: description and call of Cycle 291; Q560=1
- End of interpolation turning: description and call of Cycle 291; Q560=0
- Tool call: Recessing tool for radial recess
- Start of interpolation turning: description and call of Cycle 291; Q560=1
- End of interpolation turning: description and call of Cycle 291; Q560=0

> By converting parameter Q561, the turning tool is displayed in the simulation graphic as a milling tool.



0 BEGIN PGM 1 MM	
1 BLK FORM CYLINDER Z R15 L60	Workpiece blank definition: cylinder
2 TOOL CALL 10	Tool call: tool for axial recess
3 CC X+0 Y+0	
4 LP PR+30 PA+0 R0 FMAX	Retract the tool
5 CYCL DEF 291 COUPLG.TURNG.INTERP.	Activate interpolation turning
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
Q561=+1 ;DREHWKZ. WANDELN	
6 CYCL CALL	Call the cycle
7 LP PR+9 PA+0 RR FMAX	Position the tool in the working plane
8 L Z+10 FMAX	
9 L Z+0.2 F2000	Position the tool in the spindle axis
10 LBL 1	Recessing on level surface, infeed: 0.2 mm, depth: 6 mm
11 CP IPA+360 IZ-0.2 DR+ F10000	
12 CALL LBL 1 REP 30	

13 LBL 2	Retract from recess, step: 0.4 mm
14 CP IPA+360 IZ+0.4 DR+	
15 CALL LBL 2 REP15	
16 L Z+200 R0 FMAX	Retract to clearance height, deactivate radius compensation
17 CYCL DEF 291 COUPLG.TURNG.INTERP.	Deactivate interpolation turning
Q560=+0 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
Q561=+0 ;DREHWKZ. WANDELN	
18 CYCL CALL	Call the cycle
19 TOOL CALL 11	Tool call: tool for radial recess
20 CC X+0 Y+0	
21 LP PR+25 PA+0 R0 FMAX	Retract the tool
22 CYCL DEF 291 COUPLG.TURNG.INTERP.	Activate interpolation turning
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	
Q561=+1 ;DREHWKZ. WANDELN	
23 CYCL CALL	Call the cycle
24 LP PR+15.2 PA+0 RR FMAX	Position the tool in the working plane
25 L Z+10 FMAX	
26 L Z-11 F7000	Position the tool in the spindle axis
27 LBL 3	Recessing on lateral surface, infeed: 0.2 mm, depth: 6 mm
28 CC X+0.1 Y+0	
29 CP IPA+180 DR+ F10000	
30 CC X-0.1 Y+0	
31 CP IPA+180 DR+	
32 CALL LBL 3 REP15	
33 LBL 4	Retract from recess, step: 0.4 mm
34 CC X-0.2 Y+0	
35 CP IPA+180 DR+	
36 CC X+0.2 Y+0	
37 CP IPA+180 DR+	
38 CALL LBL 4 REP8	
39 LP PR+50 FMAX	
40 L Z+200 R0 FMAX	Retract to clearance height, deactivate radius compensation
41 CYCL DEF 291 COUPLG.TURNG.INTERP.	Deactivate interpolation turning
Q560=+0 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q216=+0 ;CENTER IN 1ST AXIS	
Q217=+0 ;CENTER IN 2ND AXIS	

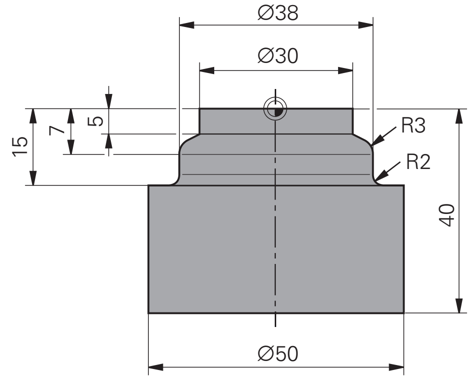
Q561=+0 ;DREHWKZ. WANDELN	
42 CYCL CALL	Call the cycle
43 TOOL CALL 11	Repeated **TOOL CALL** in order to reset the conversion of parameter Q561
44 M30	
45 END PGM 1 MM	

Example: Interpolation Turning Cycle 292

The following NC program illustrates the use of Cycle **292 CONTOUR.TURNG.INTRP.** This example illustrates the machining of an outside contour with the milling spindle rotating.

Program run

- Tool call: Milling cutter D20
- Cycle 32 Tolerance
- Reference to the contour with Cycle 14
- Cycle 292 Contour turning interpolation



0 BEGIN PGM 2 MM	
1 BLK FORM CYLINDER Z R25 L40	Workpiece blank definition: cylinder
2 TOOL CALL "D20" Z S111	Tool call: End mill D20
3 CYCL DEF 32.0 TOLERANCE	Use Cycle 32 to define the tolerance
4 CYCL DEF 32.1 T0.05	
5 CYCL DEF 32.2 HSC-MODE:1	
6 CYCL DEF 14.0 CONTOUR	Use Cycle 14 to refer to the contour in LBL1
7 CYCL DEF 14.1 CONTOUR LABEL1	
8 CYCL DEF 292 CONTOUR.TURNG.INTRP.	Define Cycle 292
Q560=+1 ;SPINDLE COUPLING	
Q336=+0 ;ANGLE OF SPINDLE	
Q546=+3 ;CHANGE TOOL DIRECTN.	
Q529=+0 ;MACHINING OPERATION	
Q221=+0 ;SURFACE OVERSIZE	
Q441=+1 ;INFEED	
Q449=+15000 ;FEED RATE	
Q491=+15 ;CONTOUR START RADIUS	
Q357=+2 ;CLEARANCE TO SIDE	
Q445=+50 ;CLEARANCE HEIGHT	
9 L Z+50 R0 FMAX M3	Pre-position in the tool axis, spindle on
10 L X+0 Y+0 R0 FMAX M99	Pre-position in the working plane to the center of rotation, call the cycle
11 LBL 1	LBL1 contains the contour
12 L Z+2 X+15	
13 L Z-5	
14 L Z-7 X+19	
15 RND R3	
16 L Z-15	
17 RND R2	
18 L X+27	

19 LBL 0	
20 M30	End of program
21 END PGM 2 MM	

Example of hob milling

The following NC program illustrates the use of Cycle 286 **GEAR HOBBING**. This programming example shows how to machine an involute spline with module=1 (deviating from DIN 3960).

Program run

- Tool call: Gear hob
- Start turning mode
- Reset the coordinate system with Cycle 801
- Move to safe position
- Define Cycle 285
- Call Cycle 286
- Reset the coordinate system with Cycle 801

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI+58	Workpiece blank definition: cylinder
2 TOOL CALL "ABWAELZFRAESER"	Call the tool
3 FUNCTION MODE TURN	Activate turning mode
4 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	Reset the coordinate system
5 M145	Deactivate M144 if still active
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	Constant surface speed OFF
7 M140 MB MAX	Retract the tool
8 L A+0 R0 FMAX	Set the rotary axis to 0
9 L X0 Y0 R0 FMAX	Pre-position the tool in the workpiece center
10 Z+50 R0 FMAX	Pre-position the tool in the spindle axis
11 CYCL DEF 285 ZAHNRAD DEFINIEREN	Define Cycle 285
Q551=+0 ;STARTING POINT IN Z	
Q552=-11 ;END POINT IN Z	
Q540=+1 ;MODULE	
Q541=+90 ;NUMBER OF TEETH	
Q542=+90 ;OUTSIDE DIAMETER	
Q563=+1 ;TOOTH HEIGHT	
Q543=+0.05 ;TROUGH-TIP CLEARANCE	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 286 ZAHNRAD WAELZFRAESEN	Define Cycle 286
Q215=+0 ;MACHINING OPERATION	
Q200=+2 ;SET-UP CLEARANCE	
Q260=+30 ;CLEARANCE HEIGHT	
Q545=+1.6 ;TOOL LEAD ANGLE	
Q546=+0 ;CHANGE ROTATION DIR.	
Q547=+0 ;ANG. OFFSET, SPINDLE	
Q550=+1 ;MACHINING SIDE	
Q533=+1 ;PREFERRED DIRECTION	
Q530=+2 ;INCLINED MACHINING	

Q253=+2222 ;F PRE-POSITIONING	
Q553=+5 ;TOOL LENGTH OFFSET	
Q554=+10 ;SYNCHRONOUS SHIFT	
Q548=+1 ;ROUGHING SHIFT	
Q463=+1 ;MAX. CUTTING DEPTH	
Q488=+0.3 ;PLUNGING FEED RATE	
Q478=+0.3 ;PLUNGING FEED RATE	
Q483=+0.4 ;OVERSIZE FOR DIAMETER	
Q505=+0.2 ;FINISHING FEED RATE	
Q549=+3 ;FINISHING SHIFT	
13 CYCL CALL M303	Call the cycle, spindle ON
14 FUNCTION MODE MILL	Activate milling mode
15 M140 MB MAX	Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	Reset the rotation
17 M30	End of program
18 END PGM 5 MM	

Example of skiving

The following NC program illustrates the use of Cycle 287 **GEAR SKIVING**. This programming example shows how to machine an involute spline with module=1 (deviating from DIN 3960).

Program run

- Tool call: Internal gear cutter
- Start turning mode
- Reset the coordinate system with Cycle 801
- Move to safe position
- Define Cycle 285
- Call Cycle 287
- Reset the coordinate system with Cycle 801

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI+58	Workpiece blank definition: cylinder
2 TOOL CALL "Hohlradfraeser"	Call the tool
3 FUNCTION MODE TURN	Activate turning mode
4 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	Reset the coordinate system
5 M145	Deactivate M144 if still active
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	Constant surface speed OFF
7 M140 MB MAX	Retract the tool
8 L A+0 R0 FMAX	Set the rotary axis to 0
9 L X0 Y0 R0 FMAX	Pre-position the tool in the workpiece center
10 Z+50 R0 FMAX	Pre-position the tool in the spindle axis
11 CYCL DEF 285 ZAHNRAD DEFINIEREN	Define Cycle 285
Q551=+0 ;STARTING POINT IN Z	
Q552=-11 ;END POINT IN Z	
Q540=+1 ;MODULE	
Q541=+90 ;NUMBER OF TEETH	
Q542=+90 ;OUTSIDE DIAMETER	
Q563=+1 ;TOOTH HEIGHT	
Q543=+0.05 ;TROUGH-TIP CLEARANCE	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 287 ZAHNRAD WAELZSCHAELEN	Define Cycle 287
Q240=+5 ;NUMBER OF CUTS	
Q584=+1 ;NO. OF FIRST CUT	
Q585=+5 ;NO. OF LAST CUT	
Q200=+2 ;SET-UP CLEARANCE	
Q260=+50 ;CLEARANCE HEIGHT	
Q545=+20 ;TOOL LEAD ANGLE	
Q546=+0 ;CHANGE ROTATION DIR.	
Q547=+0 ;ANG. OFFSET, SPINDLE	
Q550=+1 ;MACHINING SIDE	

Q533=+1 ;PREFERRED DIRECTION	
Q530=+2 ;INCLINED MACHINING	
Q253=+2222 ;F PRE-POSITIONING	
Q586=+0,4 ;FIRST INFEED	
Q587=+0,1 ;LAST INFEED	
Q588=+0,4 ;FIRST FEED RATE	
Q589=+0,25 ;LAST FEED RATE	
Q580=+0,2 ;FEED-RATE ADAPTION	
13 CYCL CALL M303	Call the cycle, spindle ON
14 FUNCTION MODE MILL	Activate milling mode
15 M140 MB MAX	Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	Reset the rotation
17 M30	End of program
18 END PGM 5 MM	

13

Cycles: Turning

13.1 Turning Cycles (software option 50)

Overview

Defining turning cycles:

CYCL DEF

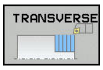
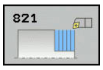
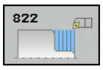
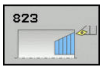
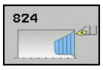
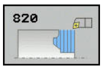
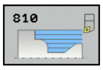
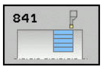
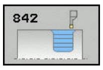
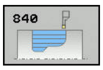
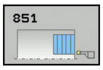
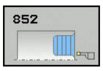
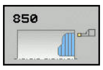
- The soft-key row shows the available groups of cycles

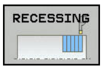
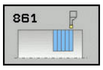
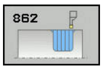
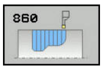
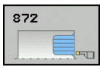
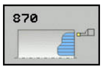
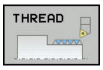
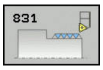
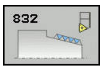
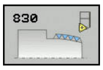
TURNING

- Menu for cycle group: Press the **Turning** soft key
- Select cycle group, e.g. cycles for longitudinal turning
- Select cycle, e.g. TURN SHOULDER, LONGITUDINAL

The control offers the following cycles for turning operations:

Soft key	Cycle group	Cycle	Page
SPECIAL CYCLES	**Special cycles**		
800		ADJUST XZ SYSTEM (Cycle 800, ISO: G800)	408
801		RESET ROTARY COORDINATE SYSTEM (Cycle 801, ISO: G801)	415
880		GEAR HOBBING (Cycle 880, ISO: G880)	516
892		CHECK UNBALANCE (Cycle 892, ISO: G892)	523
LONGITDNL	**Cycles for longitudinal turning**		417
811		TURN SHOULDER LONGITUDINAL (Cycle 811, ISO: G811)	418
812		TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, ISO: G812)	420
813		TURN, LONGITUDINAL PLUNGE (Cycle 813, ISO: G813)	423
814		TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, ISO: G814)	426
810		TURN CONTOUR LONGITUDINAL (Cycle 810, ISO: G810)	429
815		TURN CONTOUR-PARALLEL (Cycle 815, ISO: G815)	433

Soft key	Cycle group	Cycle	Page
	**Cycles for transverse turning**		417
		TURN SHOULDER FACE (Cycle 821, ISO: G821)	436
		TURN SHOULDER FACE EXTENDED (Cycle 822, ISO: G822)	438
		TURN, TRANSVERSE PLUNGE (Cycle 823, ISO: G823)	442
		TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, ISO: G824)	445
		TURN CONTOUR FACE (Cycle 820, ISO: G820)	449
		TURN CONTOUR-PARALLEL (Cycle 815, ISO: G815)	433
	**Cycles for recess turning**		
		SIMPLE RADIAL RECESSING (Cycle 841, ISO: G841)	453
		RADIAL RECESSING EXTENDED (Cycle 842, ISO: G842)	456
		RECESSING CONTOUR RADIAL (Cycle 840, ISO: G840)	460
		SIMPLE AXIAL RECESSING (Cycle 851, ISO: G851)	464
		AXIAL RECESSING EXTENDED (Cycle 852, ISO: G852)	467
		AXIAL RECESSING (Cycle 850, ISO: G850)	471

Soft key	Cycle group	Cycle	Page
	**Cycles for recessing**		
		RADIAL RECESSING (Cycle 861, ISO: G861)	475
		RADIAL RECESSING EXTENDED (Cycle 862, ISO: G862)	478
		RECESSING CONTOUR RADIAL (Cycle 860, ISO: G860)	482
		AXIAL RECESSING (Cycle 871, ISO: G871)	486
		AXIAL RECESSING EXTENDED (Cycle 872, ISO: G872)	489
		AXIAL RECESSING (Cycle 870, ISO: G870)	494
	**Cycles for thread turning**		
		THREAD LONGITUDINAL (Cycle 831, ISO: G831)	498
		THREAD EXTENDED (Cycle 832, ISO: G832)	502
		CONTOUR-PARALLEL THREAD (Cycle 830, ISO: G830)	507
	Cycle for simultaneous turning		
		TURNING SIMULTANEOUS FINISHING (Cycle 883, ISO: G883, software option #158)	511

Working with turning cycles

> You can only use turning cycles in Turning mode **FUNCTION MODE TURN**.

In turning cycles, the control takes the cutting geometry (**TO, RS, P-ANGLE, T-ANGLE**) of the tool into account in order to prevent damage to the defined contour elements. If it is not possible to machine the entire contour with the active tool, the control will display a warning.

You can use the turning cycles both for inside and outside machining. Depending upon the specific cycle, the control detects the machining position (inside/outside machining) via the starting position or tool position when the cycle is called. In some cycles you can also enter the machining position directly in the cycle. After modifying the machining position, check the tool position and the direction of rotation.

If you program **M136** before a cycle, the control interprets feed rate values in the cycle in mm/rev.; without **M136** in mm/min.

If you execute turning cycles with inclined machining (**M144**), the angles of the tool with respect to the contour change. The control automatically takes these modifications into account and thus also monitors the machining in inclined state to prevent contour damage.

Some cycles machine contours that you have written in a subprogram. You program these contours with path functions or FK functions. Before calling the cycle you must program the cycle **14 CONTOUR** to define the subprogram number.

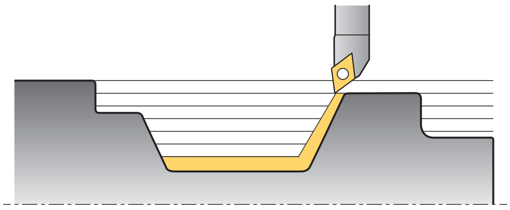
You must call turning cycles 880 and 81x - 87x with **CYCL CALL** or **M99**. Before calling a cycle, be sure to program:

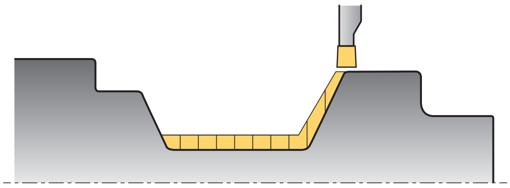
- Turning mode: **FUNCTION MODE TURN**
- Call a tool with **TOOL CALL**
- Direction of rotation of turning spindle, e.g. **M303**
- Selection of speed/cutting speed: **FUNCTION TURNDATA SPIN**
- If you use feed rate per revolution mm/rev., **M136**
- Position the tool to a suitable starting point e.g. **L X+130 Y +0 R0 FMAX**
- Adapt the coordinate system and align the tool: **CYCL DEF 800 ADJUST XZ SYSTEM**

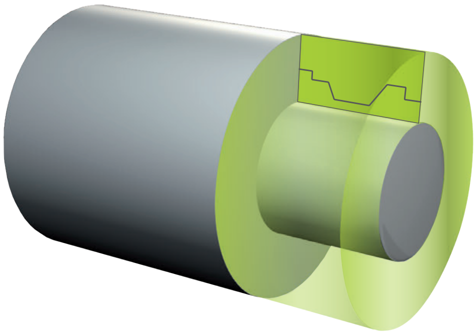
Blank form update (FUNCTION TURNDATA)

During turning operations workpieces must often be machined with several tools. Often a contour element cannot be completely finished because the tool shape does not permit this (e.g. with an undercut). In this case, single sub-areas have to be reworked with other tools. Using the contour follow-up function, the control detects the already machined areas and adapts all approach and departure paths to the specific, current machining situation. With the shorter machining paths, air cuts are avoided to significantly reduce machining time.

To activate contour follow-up, program the **TURNDATA BLANK** function and reference an NC program or subprogram with a workpiece blank specification. The workpiece blank defined in **TURNDATA BLANK** determines the area to be machined with the contour follow-up. **TURNDATA BLANK OFF** deactivates the contour follow-up.







NOTICE

Danger of collision!

Contour follow-up is used to optimize machining areas and approach movements. For approach and departure paths, the control takes the specific workpiece blank into account that is being followed. If parts of the finished part extend beyond the workpiece blank, this may damage the workpiece and tool.

- Define the workpiece blank larger than the finished part

Blank form update is only possible with cycle machining in turning mode (**FUNCTION MODE TURN**).

You must define a closed contour as the workpiece blank for the blank form update (start position = end position). The workpiece blank corresponds to the cross-section of a rotationally symmetrical body.

The control provides various options for defining the workpiece blank:

Soft key	Workpiece blank definition
BLANK OFF	Deactivate blank form update **TURNDATA BLANK OFF**: No input
BLANK <FILE>	Workpiece blank definition in an NC program: Enter the name of the file
BLANK <FILE>=QS	Workpiece blank definition in an NC program: Enter the string parameter with the program name
BLANK LBL NR	Workpiece blank definition in a subprogram: Enter the number of the subprogram
BLANK LBL NAME	Workpiece blank definition in a subprogram: Enter the name of the subprogram

Soft key	Workpiece blank definition
BLANK LBL QS	Workpiece blank definition in a subprogram: Enter the string parameter with the subprogram name

Activate blank form update and define workpiece blank:

SPEC FCT
- Show the soft-key row with special functions

TURNING PROGRAM FUNCTIONS
- Menu: Press the **TURNING PROGRAM FUNCTIONS** soft key

FUNCTION TURNDATA
- Press the **BASIC FUNCTIONS** soft key

TURNDATA BLANK
- Select the function for blank form update

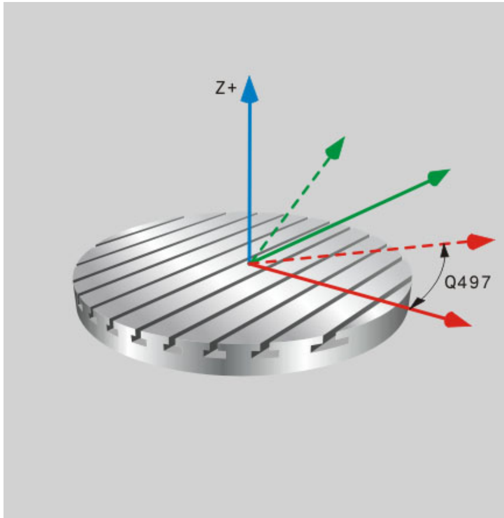
Example

```
11 FUNCTION TURNDATABLANK LBL 20
```

13.2 ADJUST XZ SYSTEM (Cycle 800, ISO: G800)

Application

This function must be adapted by your machine manufacturer.



In order to be able to perform a turning operation, you need to position the tool appropriately with respect to the turning spindle. You can use Cycle **800 ADJUST XZ SYSTEM** for this purpose.

With turning operations, the inclination angle between the tool and turning spindle is important, in order to e.g. machine contours with undercuts. Cycle 800 provides various possibilities for aligning the coordinate system for an inclined machining operation:

- If you have positioned the tilting axis for inclined machining, you can use Cycle 800 to orient the coordinate system to the positions of the tilting axes (**Q530=0**)
- Cycle 800 uses the inclination angle Q531 to calculate the required tilting axis angle. Depending on the strategy selected in parameter **INCLINED MACHINING Q530**, the control positions the tilting axis with (**Q530=1**) or without compensation movement (**Q530=2**)
- Cycle 800 uses the inclination **angle Q531** to calculate the required tilting axis angle, but does not position the tilting axis (**Q530=3**). You need to position the tilting axis after the cycle manually to the calculated values Q120 (A axis), Q121 (B axis), and Q122 (C axis).

If you change the position of a tilting axis, you need to run Cycle 800 again to align the coordinate system.

If the milling spindle axis and the turning spindle axis are parallel to each other, you can use the **Precession angle Q497** to define any desired rotation of the coordinate system about the spindle axis (Z axis). This may be necessary if you have to bring the tool into a specific position due to space restrictions or if you want to improve your ability to observe a machining process. If the turning spindle and milling spindle axes are not parallel, only two precession angles are realistic for machining. The control selects the angle that is closest to the input value of **Q497**.

Cycle 800 positions the milling spindle such that the cutting edge is aligned relative to the turning contour. You can use a mirrored version of the tool (**REVERSE TOOL Q498**); this offsets the milling spindle by 180°. In this way, you can use your tools both for inside and outside machining. Position the cutting edge at the center of the turning spindle using a positioning block, such as **L Y+0 R0 FMAX**.

Eccentric turning

Sometimes it is not possible to clamp a workpiece such that the axis of rotation is aligned with the axis of the turning spindle. For example, this is the case with large or rotationally non-symmetric workpieces. The eccentric turning **Q535** function in Cycle 800 enables you to perform turning such cases as well.

During eccentric turning, more than one linear axis is coupled to the turning spindle. The control compensates the eccentricity by performing circular compensating movements with the coupled linear axes.

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

If you machine with high speed and a high amount of eccentricity, you need to program large feed rates for the linear axes in order to perform the movements synchronously. If these feed rates are not met, the contour would be damaged. The control therefore generates an error message if 80 % of a maximum axis speed or acceleration is exceeded. If this occurs, reduce the speed.

NOTICE

Danger of collision!

The Control performs compensating movements during coupling and decoupling. Check for possible collisions.

- Coupling and decoupling must be performed while the spindle is stationary.

NOTICE

Danger of collision!

Collision monitoring (DCM) is not active during eccentric turning. The control displays a corresponding warning during eccentric turning.

- Watch out for possible collisions!

NOTICE

Danger of collision!

The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.

- Select the specifications in such a way that no vibrations (resonances) occur.

Turn a test cut before the actual machining operation to ensure that the required speeds can be attained.

The linear axis positions resulting from the compensation are displayed by the control only in the ACTUAL value position display.

Effect

With Cycle 800 **ADJUST XZ SYSTEM**, the control aligns the workpiece coordinate system and orients the tool correspondingly. Cycle 800 is effective until it is reset by Cycle 801, or until Cycle 800 is defined again. Some cycle functions of Cycle 800 are implicitly reset by other factors:

- Mirroring of tool data (**Q498 REVERSE TOOL**) is reset by a tool call with **TOOL CALL**.
- The **ECCENTRIC TURNING Q535** function is reset at the end of the program or if the program is aborted (internal stop).

Please note while programming:

Refer to your machine manual!

Cycle 800 **ADJUST XZ SYSTEM** is machine-dependent. Refer to your machine manual!

Software option 50 must be enabled.

Software option 135 must be enabled.

The machine tool builder determines the configuration of your machine. In this configuration if the tool spindle is defined as the axis in the kinematics, the feed-rate potentiometer is effective with movements with Cycle 800.

The machine tool builder can determine how exactly the precession angle aligns the tool.

The tool must be clamped and measured in the correct position.

You can mirror the tool data (**Q498 REVERSE TOOL**) only if a turning tool has been selected.

Check the orientation of the tool before machining.

To reset Cycle 800, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM**.

Cycle 800 limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You may have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle 800. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle 800, the smaller value will be applied. To reset Cycle 800, program Cycle 801. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** applies again.

If you use the 1: MOVE, 2: TURN, and 3: STAY settings in parameter **Q530 Inclined machining**, the control will activate (depending on the machine configuration) the **M144** function or TCPM. (**For more information:** User's Manual for Setup, Testing and Running NC programs.)

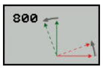
NOTICE

Danger of collision!

If **Q498**=1 and you additionally program the **FUNCTION LIFTOFF ANGLE TCS** function, there might be different results, depending on the configuration. If the tool spindle has been defined as an axis, the **LIFTOFF** will be included in the rotation when reversing the tool. If the tool spindle has been defined as a kinematic transformation, the **LIFTOFF** will **not** be included in the rotation when reversing the tool!

- Carefully test the NC program or program section in **Program run, single block** operating mode.
- If required, change the algebraic sign of the SPB angle.

Cycle parameters



- **Q497 Precession angle?**: Angle to which the control aligns the tool. Input range: 0 to 359.9999
- **Q498 Reverse tool (0=no/1=yes)?**: Mirror tool for inside/outside machining. Input range: 0 or 1.
- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  0: Maintain position of tilted axes (axis must already have been positioned)
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN)
  3: Do not position the tilting axis. Position the tilting axes later in a separate positioning block (STAY). The control stores the position values in the parameters Q120 (A axis), Q121 (B axis) and Q122 (C axis).
- **Q531 Angle of incidence?**: Angle of incidence for aligning the tool. Input range: -180° to +180°
- **Q532 Feed rate for positioning?**: Traverse speed of the tilting axis during automatic positioning. Input range: 0.001 to 99999.999

- **Q533 Preferred dir. of incid. angle?**: Selection of alternative inclination options. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on the machine. In general, there are always two possible solutions. Using parameter Q533, configure which of the solution options the control should apply:
  0: Option with the shortest distance from the current position
  -1: Option in the range between 0° and -179.9999°
  +1: Option in the range between 0° and +180°
  -2: Option in the range between -90° and -179.9999°
  +2: Option in the range between +90° and +180°
- **Q535 Eccentric turning?**: Couple the axes for the eccentric turning operation:
  0: Deactivate axis couplings
  1: Activate axis couplings The center of rotation is located at the active Preset
  2: Activate axis couplings. The center of rotation is located at the active datum
  3: Do not change axis couplings
- **Q536 Eccentric turning without stop?**: Interrupt program run before the axes are coupled:
  0: Stop before the axes are coupled again. In stopped condition, the control opens a window in which the amount of eccentricity and the maximum deflection of the individual axes are displayed. Then press **NC start** to resume machining or press the **CANCEL** soft key to cancel machining
  1: Axes are coupled without stopping beforehand

13.3 RESET ROTARY COORDINATE SYSTEM (Cycle 801, ISO: G801)

Please note while programming:

Cycle **RESET ROTARY COORDINATE SYSTEM** is machine-dependent. Refer to your machine manual!

With Cycle 801 **RESET ROTARY COORDINATE SYSTEM**, you can reset the settings you have made with Cycle 800 **ADJUST XZ SYSTEM**.

To reset Cycle 800, program Cycle 801 **RESET ROTARY COORDINATE SYSTEM**.

Cycle 800 limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You may have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle 800. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle 800, the smaller value will be applied. To reset Cycle 800, program Cycle 801. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** applies again.

Effect

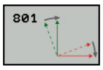
Cycle 801 resets the following settings you have programmed with Cycle 800:

- Precession angle Q497
- Reverse tool Q498

If you have executed the eccentric turning function with Cycle 800, please note the following: Cycle 800 limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You may have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle 800. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle 800, the smaller value will be applied. To reset Cycle 800, program Cycle 801. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** applies again.

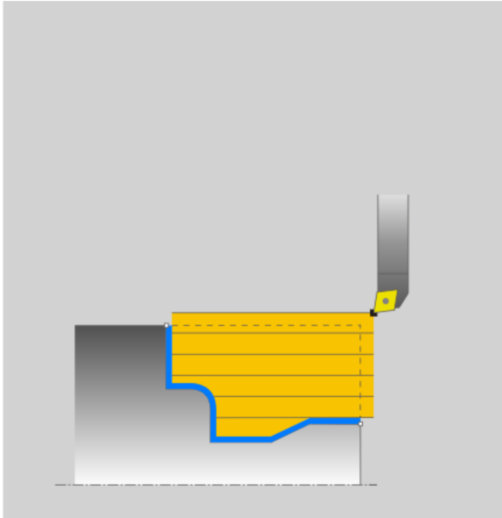
Cycle 801 does not orient the tool to the starting position. If a tool was oriented with Cycle 800, it remains in this position also after resetting.

Cycle parameters



- Cycle 801 does not have a cycle parameter.
  Finish the cycle input with the **END** key.

13.4 Fundamentals of Turning Cycles



The pre-positioning of the tool has a decisive influence on the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when the cycle is called. When calculating the area to be machined, the control takes into account the starting point and the end point defined in the cycle or of contour defined in the cycle. If the starting point lies in the area to be machined, the control positions the tool to set-up clearance beforehand in some cycles.

The turning direction with 81x cycles is longitudinal to the rotary axis and lateral to the rotary axis with 82x cycles. The motions are contour-parallel in cycle 815.

The cycles can be used for inside and outside machining. The Control takes the information for this from the position of the tool or the definition of the cycle (see "Working with turning cycles", Page 405).

In cycles with freely defined contours (Cycles 810, 820 and 815), the programming direction of the contour determines the direction of machining.

In cycles for turning you can specify the machining strategies of roughing, finishing or complete machining.

NOTICE

Danger of collision!

The turning cycles position the tool automatically to the starting point during finishing. The approach strategy is influenced by the position of the tool when the cycle is called. The decisive factor is whether the tool is located inside or outside an envelope contour when the cycle is called. The envelope contour is the programmed contour, enlarged by the set-up clearance. If the tool is within the envelope contour, the cycle positions the tool at the defined feed rate directly to the starting position. This can cause contour damage.

- Position the tool at a sufficient distance from the starting point to prevent the possibility of contour damage
- If the tool is outside the envelope contour, positioning to the envelope contour is performed at rapid traverse, and at the programmed feed rate within the envelope contour.

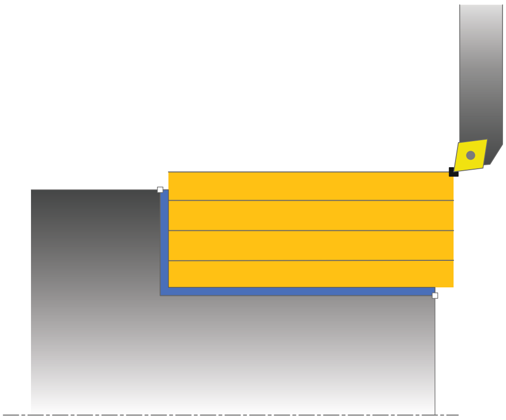
13.5 TURN SHOULDER LONGITUDINAL (Cycle 811, ISO: G811)

Application

This cycle enables you to carry out longitudinal turning of right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The cycle processes the area from the tool position to the end point defined in the cycle.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 The control performs a paraxial infeed movement at rapid traverse.
3 The control finishes the contour of the finished part at the defined feed rate Q505.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

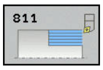
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

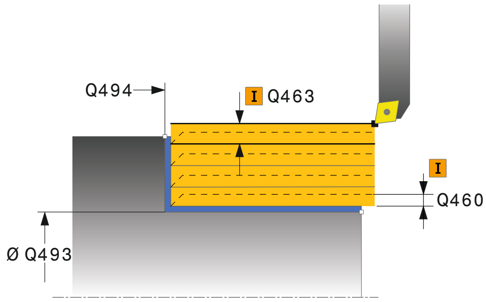
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

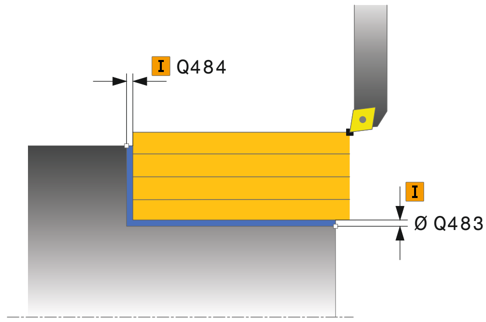
Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°





Example

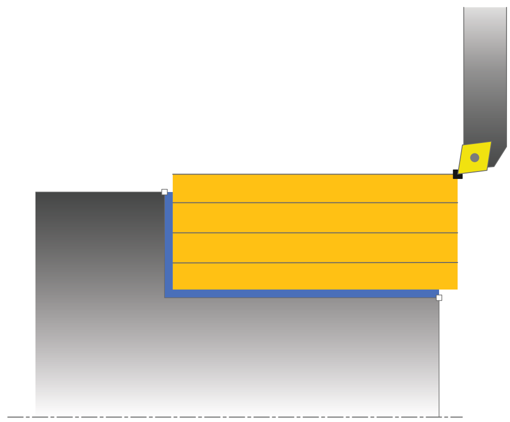
11 CYCL DEF 811 SHOULDER, LONGITDNL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-55	;CONTOUR END IN Z
Q463=+3	;MAX. CUTTING DEPTH
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q506=+0	;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

13.6 TURN SHOULDER LONGITUDINAL EXTENDED (Cycle 812, ISO: G812)

Application

This cycle enables you to run longitudinal turning of shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge



You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter Q491 is less than the end diameter Q493, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the X coordinate and then in the Z coordinate to set-up clearance and starts the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the starting point lies in the area to be machined, the control positions the tool to set-up clearance beforehand.

1 The control performs a paraxial infeed movement at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate Q505.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

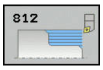
Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

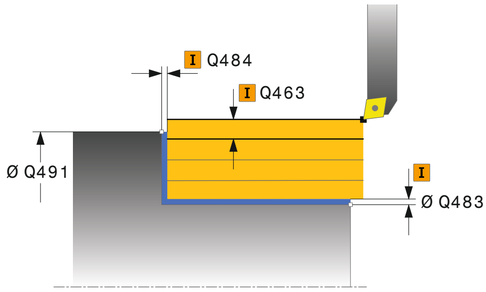
The tool position at cycle call (cycle start point) influences the area to be machined.

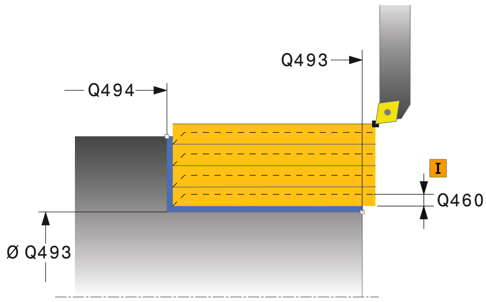
Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of circumferen. surface?**: Angle between the circumferential surface and the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q496 Angle of face?**: Angle between the level surface and the rotary axis





Example

11 CYCL DEF 812 SHOULDER, LONG. EXT.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-55	;CONTOUR END IN Z

- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Q495=+5	;ANGLE OF CIRCUM. SURFACE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+0	;ANGLE OF FACE
Q503=+1	;TYPE OF END ELEMENT
Q504=+0.5	;SIZE OF END ELEMENT
Q463=+3	;MAX. CUTTING DEPTH
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q506=+0	;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

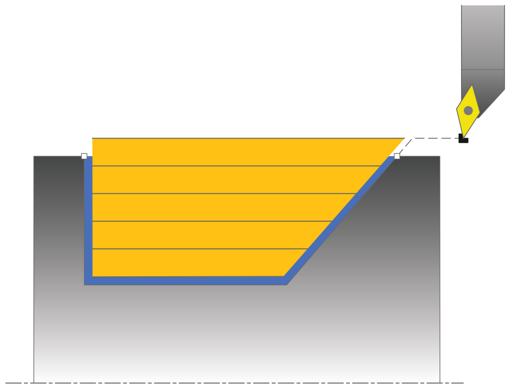
13.7 TURN, LONGITUDINAL PLUNGE (Cycle 813, ISO: G813)

Application

This cycle enables you to run longitudinal turning of shoulders with plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

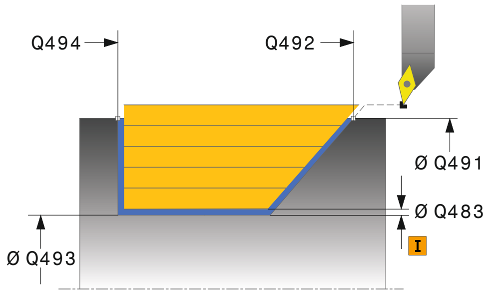
The tool position at cycle call (cycle start point) influences the area to be machined.

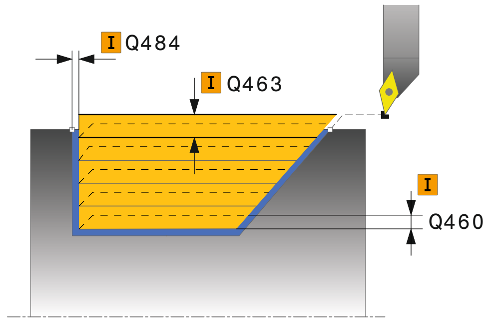
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging edge. This angle references a line perpendicular to the rotary axis.
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°



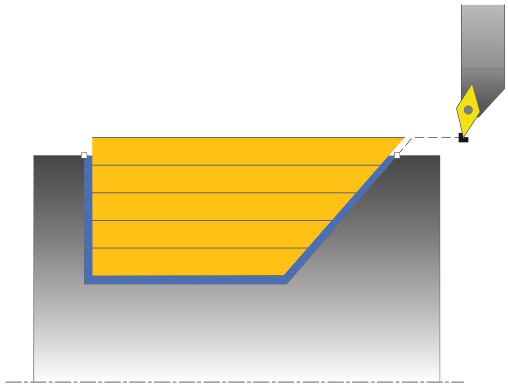


Example

11 CYCL DEF 813 TURN PLUNGE CONTOUR LONGITUDINAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-10	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-55	;CONTOUR END IN Z
Q495=+70	;ANGLE OF SIDE
Q463=+3	;MAX. CUTTING DEPTH
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q506=+0	;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

13.8 TURN, LONGITUDINAL PLUNGE EXTENDED (Cycle 814, ISO: G814)

Application



This cycle enables you to run longitudinal turning of shoulders with plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1. The infeed movement is performed at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

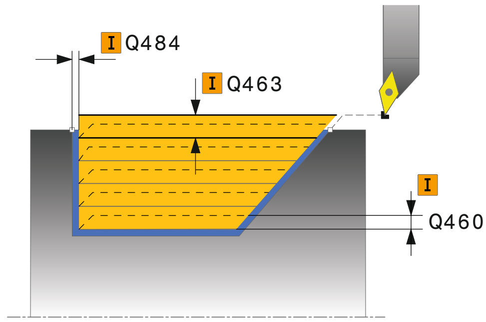
The tool position at cycle call (cycle start point) influences the area to be machined.

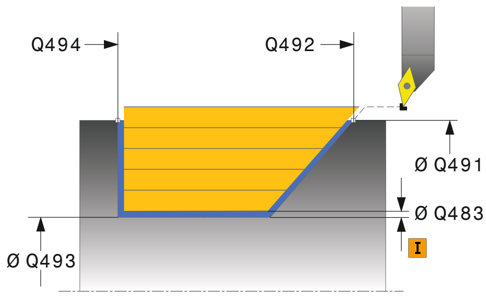
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging edge. This angle references a line perpendicular to the rotary axis.
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q496 Angle of face?**: Angle between the level surface and the rotary axis





Example

11 CYCL DEF 814 TURN PLUNGE LONGITUDINAL EXT.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-10	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-55	;CONTOUR END IN Z

- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

```
Q495=+70   ;ANGLE OF SIDE
Q501=+1    ;TYPE OF STARTING ELEMENT
Q502=+0.5  ;SIZE OF STARTING ELEMENT
Q500=+1.5  ;RADIUS OF CONTOUR EDGE
Q496=+0    ;ANGLE OF FACE
Q503=+1    ;TYPE OF END ELEMENT
Q504=+0.5  ;SIZE OF END ELEMENT
Q463=+3    ;MAX. CUTTING DEPTH
Q478=+0.3  ;ROUGHING FEED RATE
Q483=+0.4  ;OVERSIZE FOR DIAMETER
Q484=+0.2  ;OVERSIZE IN Z
Q505=+0.2  ;FINISHING FEED RATE
Q506=+0    ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

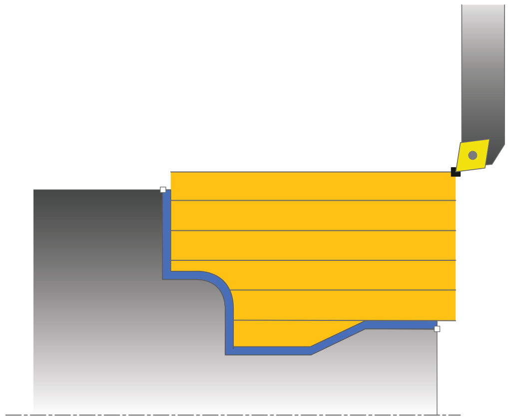
13.9 TURN CONTOUR LONGITUDINAL (Cycle 810, ISO: G810)

Application

This cycle enables you to run longitudinal turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction. The longitudinal cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

- Before the cycle is called position the tool so that it is already on the side of the cutting limit on which the material is to be cut

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle start point) influences the area to be machined.

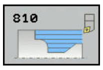
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

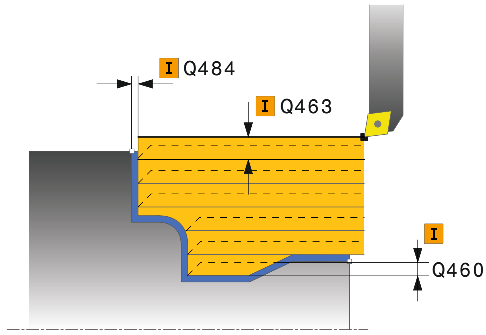
Also refer to the fundamentals of turning cycles (see Page 417).

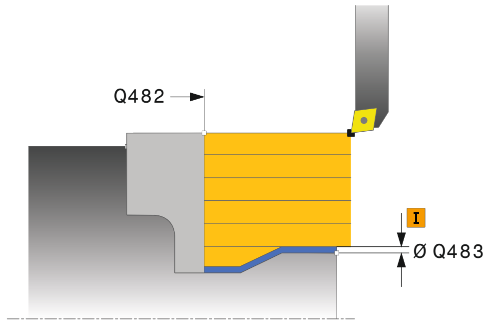
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
  2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999





- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q487 Allow plunging (0/1)?**: Permit machining of plunging elements:
  0: Do not machine plunging elements
  1: Machine plunging elements
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 810 TURN CONTOUR
   LONG.
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q499=+0     ;REVERSE CONTOUR
   Q463=+3     ;MAX. CUTTING DEPTH
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q487=+1     ;PLUNGE
   Q488=+0     ;PLUNGING FEED RATE
   Q479=+0     ;CONTOUR MACHINING
                LIMIT
   Q480=+0     ;DIAMETER LIMIT VALUE
   Q482=+0     ;LIMIT VALUE Z
   Q506=+0     ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L Z-10
18 RND R5
19 L X+40 Z-35
20 RND R5
21 L X+50 Z-40
22 L Z-55
23 CC X+60 Z-55
24 C X+60 Z-60
25 L X+100
26 LBL 0
```

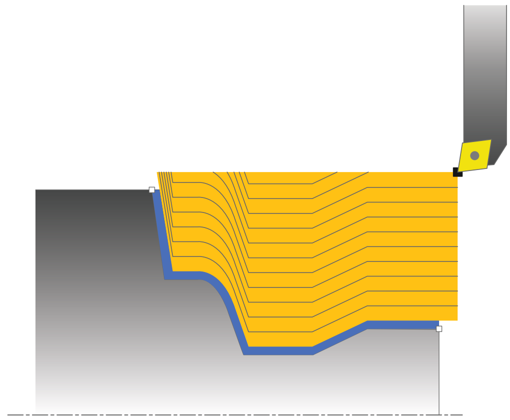
13.10 TURN CONTOUR-PARALLEL (Cycle 815, ISO: G815)

Application

This cycle enables you to run turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning with roughing is contour-parallel.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and end point. The cut is performed in contour-parallel mode at the defined feed rate **Q478**.
3 The control returns the tool at the defined feed rate back to the starting position in the X coordinate.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle start point) influences the area to be machined.

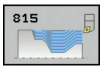
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

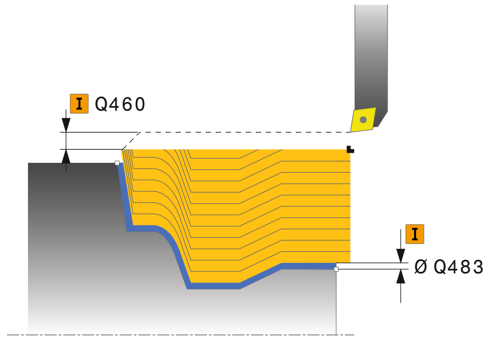
Also refer to the fundamentals of turning cycles (see Page 417).

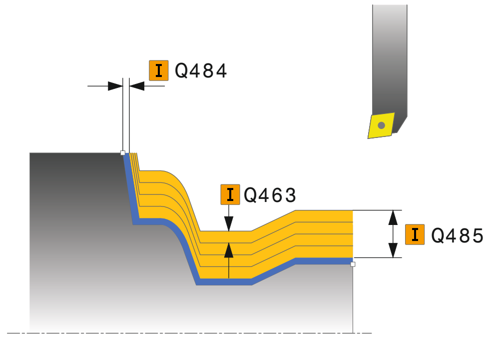
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q485 Allowance for workpiece blank?** (incremental): Contour-parallel oversize for the defined contour
- **Q486 Type of cut lines (=0/1)?**: Define the type of cutting lines:
  0: Cuts with constant chip cross section
  1: Equidistant distribution of the cuts
- **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
  2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.





Example

```
11 CYCL DEF 815  CONTOUR-PAR.
   TURNING
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q485=+5      ;ALLOWANCE ON BLANK
   Q486=+0      ;INTERSECTING LINES
   Q499=+0      ;REVERSE CONTOUR
   Q463=+3      ;MAX. CUTTING DEPTH
   Q478=0.3     ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

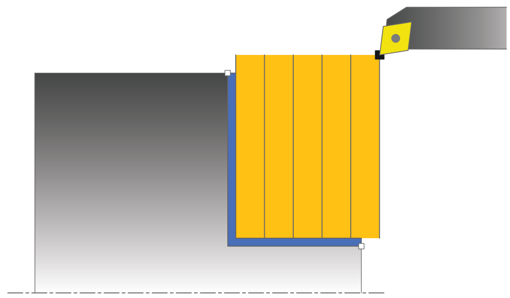
13.11 TURN SHOULDER FACE (Cycle 821, ISO: G821)

Application

This cycle enables you to face turn right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 The control performs a paraxial infeed movement at rapid traverse.
3 The control finishes the contour of the finished part at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

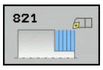
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

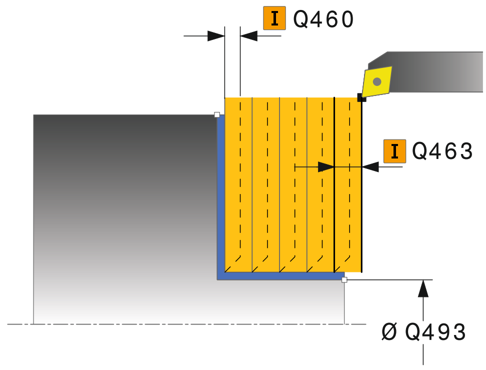
The tool position at cycle call (cycle start point) influences the area to be machined.

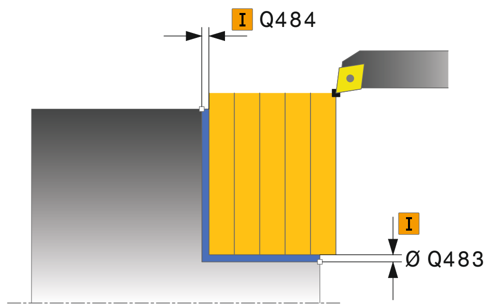
Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°





Example

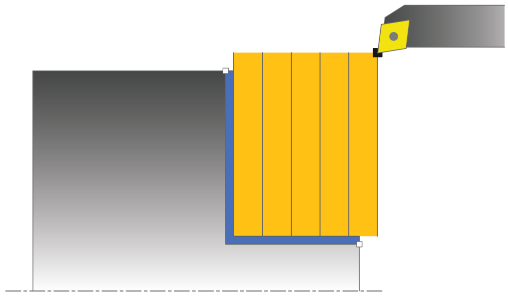
```
11 CYCL DEF 821  SHOULDER, FACE
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q493=+30    ;DIAMETER AT CONTOUR
                END
   Q494=-5     ;CONTOUR END IN Z
   Q463=+3     ;MAX. CUTTING DEPTH
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q506=+0     ;CONTOUR SMOOTHING
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

13.12 TURN SHOULDER FACE EXTENDED (Cycle 822, ISO: G822)

Application

This cycle enables you to face turn shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge



You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the Z coordinate and then in the X coordinate to set-up clearance and begins the cycle there.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1. The control performs a paraxial infeed movement at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

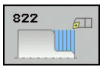
Please note while programming:

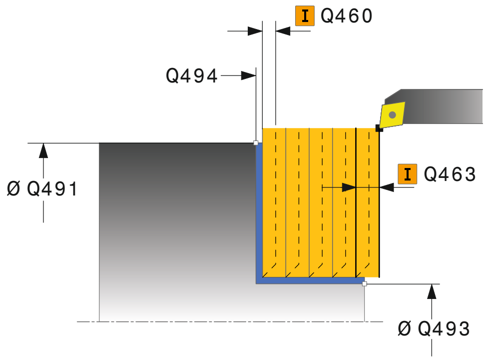
Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

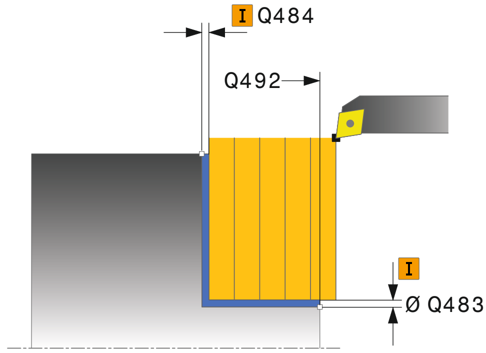
The tool position at cycle call (cycle start point) influences the area to be machined.

Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters







- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of the face?**: Angle between the level surface and the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q496 Angle of circumferen. surface?**: Angle between the circumferential surface and the rotary axis
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

11 CYCL DEF 822 SHOULDER, FACE. EXT.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+30	;DIAMETER AT CONTOUR END
Q494=-15	;CONTOUR END IN Z
Q495=+0	;ANGLE OF FACE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+5	;ANGLE OF CIRCUM. SURFACE
Q503=+1	;TYPE OF END ELEMENT
Q504=+0.5	;SIZE OF END ELEMENT
Q463=+3	;MAX. CUTTING DEPTH
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q506=+0	;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

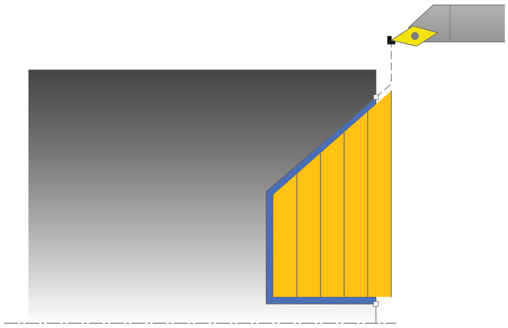
13.13 TURN, TRANSVERSE PLUNGE (Cycle 823, ISO: G823)

Application

This cycle enables you to run face turning of plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.



Roughing cycle run

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3 The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

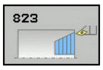
Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle start point) influences the area to be machined.

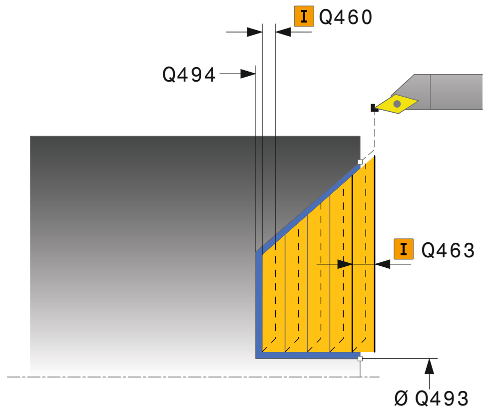
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

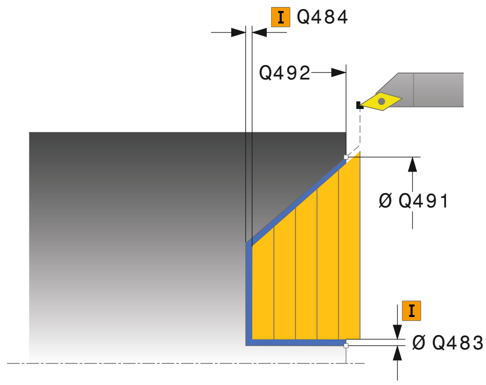
Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°





Example

11 CYCL DEF 823 TURN TRANSVERSE PLUNGE	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+20	;DIAMETER AT CONTOUR END
Q494=-5	;CONTOUR END IN Z
Q495=+60	;ANGLE OF SIDE
Q463=+3	;MAX. CUTTING DEPTH
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q506=+0	;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

13.14 TURN, TRANSVERSE PLUNGE EXTENDED (Cycle 824, ISO: G824)

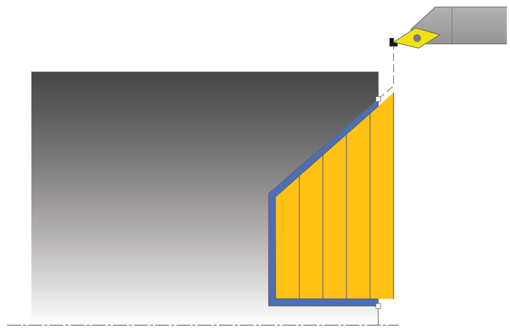
Application

This cycle enables you to run face turning of plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.



Roughing cycle run

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3. The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The infeed movement is performed at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

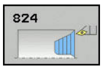
Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

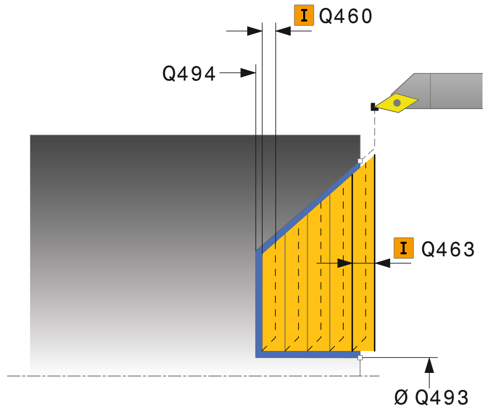
The tool position at cycle call (cycle start point) influences the area to be machined.

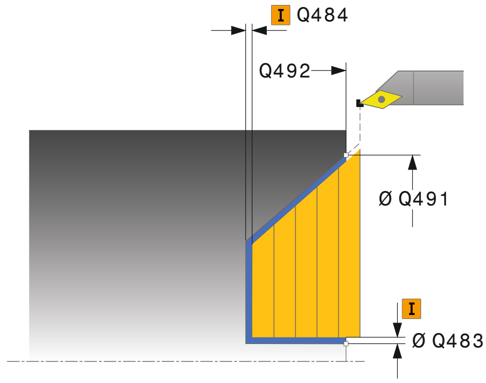
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

Also refer to the fundamentals of turning cycles (see Page 417).

Cycle parameters







- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q491 Diameter at contour start?**: X coordinate of the starting point for the plunging path (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point for the plunging path
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle of the plunging side. The reference angle is formed by the parallel line to the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius

- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour (level surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

11 CYCL DEF 824 TURN PLUNGE TRANSVERSE EXT.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=+0	;CONTOUR START IN Z
Q493=+20	;DIAMETER AT CONTOUR END
Q494=-10	;CONTOUR END IN Z
Q495=+70	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+0	;ANGLE OF FACE
Q503=+1	;TYPE OF END ELEMENT
Q504=+0.5	;SIZE OF END ELEMENT
Q463=+3	;MAX. CUTTING DEPTH
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q506=+0	;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

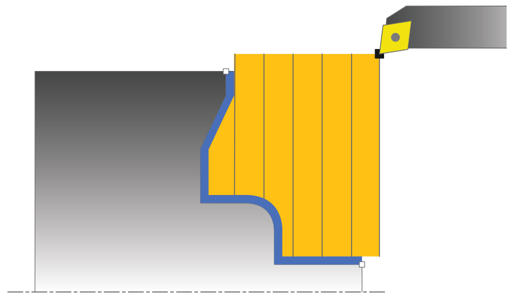
13.15 TURN CONTOUR FACE (Cycle 820, ISO: G820)

Application

This cycle enables you to run face turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction. The transverse cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

- Before the cycle is called position the tool so that it is already on the side of the cutting limit on which the material is to be cut

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The tool position at cycle call (cycle start point) influences the area to be machined.

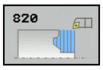
The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

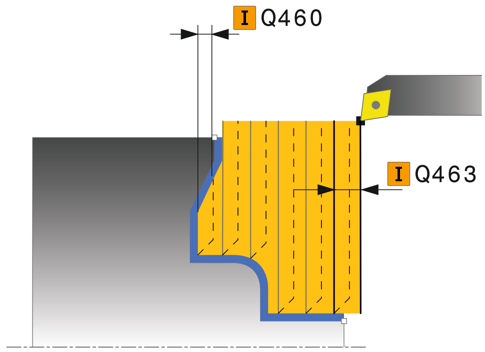
Also refer to the fundamentals of turning cycles (see Page 417).

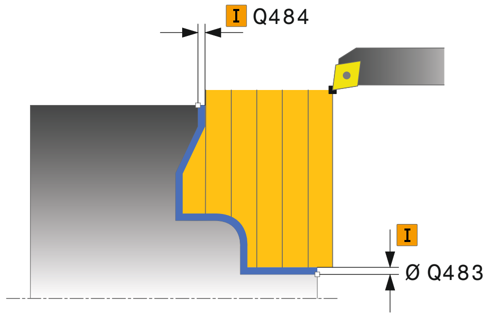
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
  2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- **Q463 Maximum cutting depth?**: Maximum infeed in axial direction. The infeed is distributed evenly to avoid abrasive cuts.





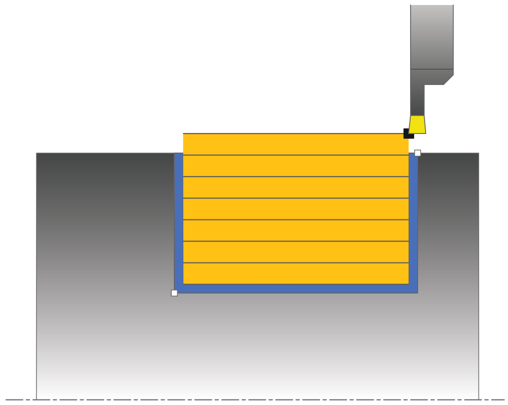
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q487 Allow plunging (0/1)?**: Permit machining of plunging elements:
  0: Do not machine plunging elements
  1: Machine plunging elements
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q506 Contour smoothing (0/1/2)?**:
  0: After each cut along the contour (within the infeed range)
  1: Contour smoothing after the last cut (entire contour); retract by 45°
  2: No contour smoothing; retract by 45°

Example

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 820 TURN CONTOUR TRANSV.
   Q215=+0      ;MACHINING OPERATION
   Q460=+2      ;SAFETY CLEARANCE
   Q499=+0      ;REVERSE CONTOUR
   Q463=+3      ;MAX. CUTTING DEPTH
   Q478=+0.3    ;ROUGHING FEED RATE
   Q483=+0.4    ;OVERSIZE FOR DIAMETER
   Q484=+0.2    ;OVERSIZE IN Z
   Q505=+0.2    ;FINISHING FEED RATE
   Q487=+1      ;PLUNGE
   Q488=+0      ;PLUNGING FEED RATE
   Q479=+0      ;CONTOUR MACHINING LIMIT
   Q480=+0      ;DIAMETER LIMIT VALUE
   Q482=+0      ;LIMIT VALUE Z
   Q506=+0      ;CONTOUR SMOOTHING
12 L X+75 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+75 Z-20
17 L X+50
18 RND R2
19 L X+20 Z-25
20 RND R2
21 L Z+0
22 LBL 0
```

13.16 SIMPLE RADIAL RECESSING (Cycle 841, ISO: G841)

Application



This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

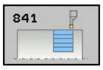
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

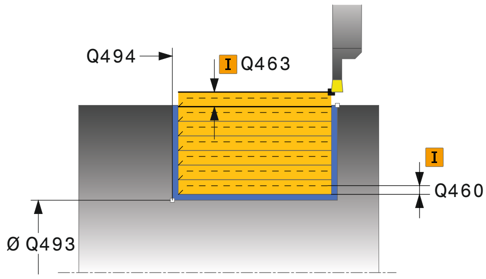
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

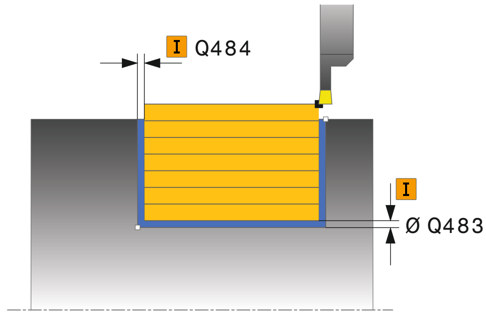
From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.





Example

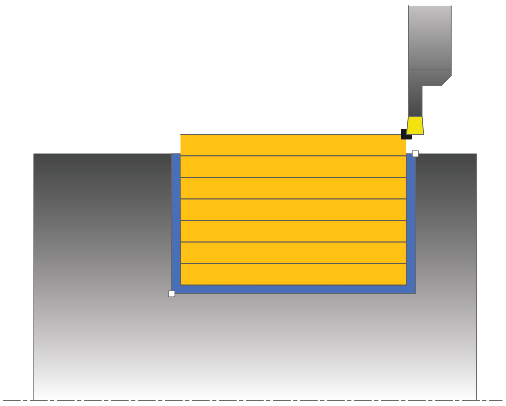
11 CYCL DEF 841 SIMPLE REC. TURNG., RADIAL DIR.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q463=+2	;MAX. CUTTING DEPTH
Q507=+0	;MACHINING DIRECTION
Q508=+0	;OFFSET WIDTH
Q509=+0	;DEPTH COMPENSATION
Q488=+0	;PLUNGING FEED RATE
12 L X+75 Y+0 Z-25 FMAX M303	
13 CYCL CALL	

13.17 RADIAL RECESSING EXTENDED (Cycle 842, ISO: G842)

Application

This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges



You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q491 Diameter at contour start**, the control positions the tool in the Z coordinate to **Q491** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q491 Diameter at contour start**, the control positions the tool in the Z coordinate to **Q491** and begins the cycle there.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate Q505.
3. The control finishes the slot floor at the defined feed rate. If a radius for contour edges Q500 was specified, the control finishes the entire slot in one pass.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

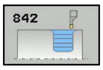
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

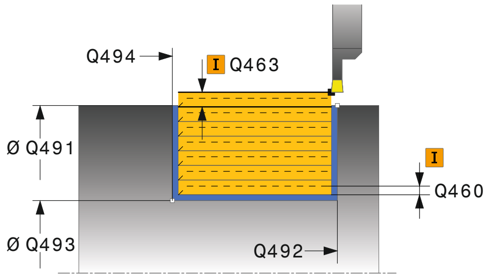
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

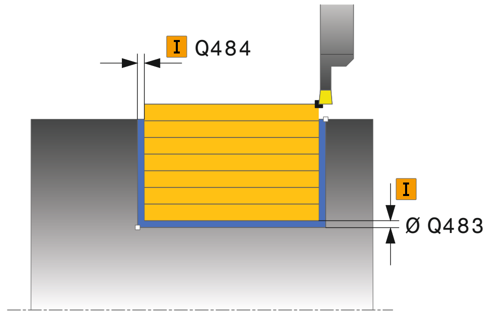
From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the perpendicular to the rotary axis.
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the perpendicular to the rotary axis.
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)





Example

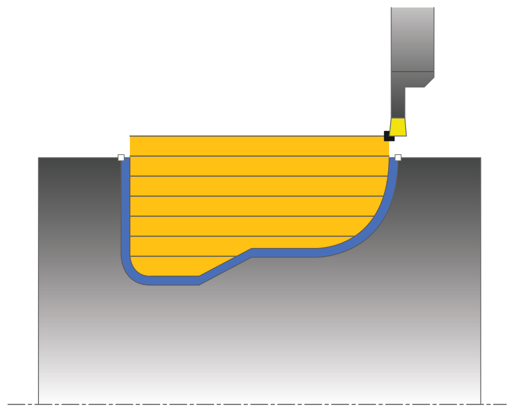
11 CYCL DEF 842 EXPND. RECESS, RADL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+5	;ANGLE OF SECOND SIDE

- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

```
    Q503=+1     ;TYPE OF END ELEMENT
    Q504=+0.5   ;SIZE OF END ELEMENT
    Q478=+0.3   ;ROUGHING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
    Q463=+2     ;MAX. CUTTING DEPTH
    Q507=+0     ;MACHINING DIRECTION
    Q508=+0     ;OFFSET WIDTH
    Q509=+0     ;DEPTH COMPENSATION
    Q488=+0     ;PLUNGING FEED RATE
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

13.18 RECESSING CONTOUR RADIAL (Cycle 840, ISO: G840)

Application



This cycle enables you to recess slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the X coordinate of the starting point is less than the contour starting point, the control positions the tool in the X coordinate to the contour starting point and begins the cycle there.

1. The control positions the tool at rapid traverse in the Z coordinate (first recessing position).
2. The control performs a recessing traverse until the first plunging depth is reached.
3. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
4. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6. The tool recesses to the next plunging depth.
7. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side walls of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

- Before the cycle is called position the tool so that it is already on the side of the cutting limit on which the material is to be cut

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

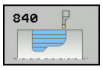
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

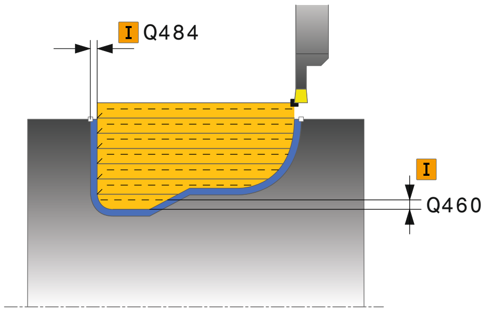
Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

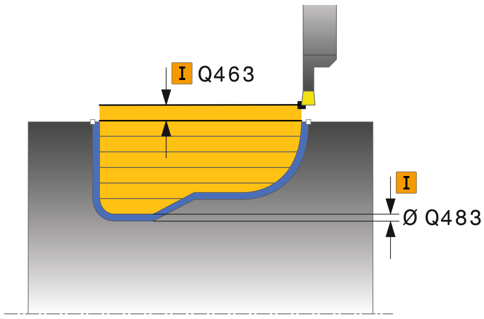
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

Cycle parameters







- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction

- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q499 Reverse contour (0=no/1=yes)?**: Machining direction:
  0: Contour machined in the programmed direction
  1: in reverse direction to the programmed direction

Example

```
9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 840 RECESS TURNG,
   RADIAL
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q478=+0.3   ;ROUGHING FEED RATE
   Q488=+0     ;PLUNGING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
   Q479=+0     ;CONTOUR MACHINING
               LIMIT
   Q480=+0     ;DIAMETER LIMIT VALUE
   Q482=+0     ;LIMIT VALUE Z
   Q463=+2     ;MAX. CUTTING DEPTH
   Q507=+0     ;MACHINING DIRECTION
   Q508=+0     ;OFFSET WIDTH
   Q509=+0     ;DEPTH COMPENSATION
   Q499=+0     ;REVERSE CONTOUR
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z-10
17 L  X+40 Z-15
18 RND  R3
19 CR  X+40 Z-35  R+30 DR+
18 RND  R3
20 L  X+60 Z-40
21 LBL  0
```

13.19 SIMPLE AXIAL RECESSING (Cycle 851, ISO: G851)

Application

This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1 From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate Q505.
7 The control returns the tool at rapid traverse to the cycle starting point.

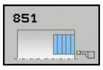
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

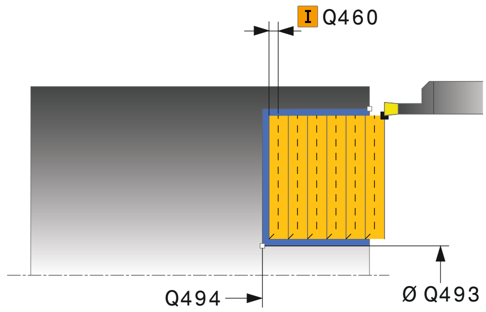
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

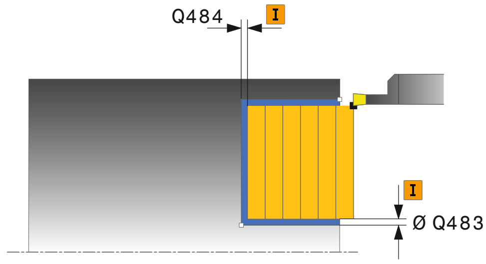
From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.



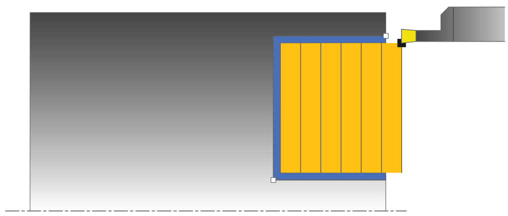


Example

```
11 CYCL DEF 851 SIMPLE REC TURNG,
   AX
   Q215=+0    ;MACHINING OPERATION
   Q460=+2    ;SAFETY CLEARANCE
   Q493=+50   ;DIAMETER AT CONTOUR
              END
   Q494=-10   ;CONTOUR END IN Z
   Q478=+0.3  ;ROUGHING FEED RATE
   Q483=+0.4  ;OVERSIZE FOR DIAMETER
   Q484=+0.2  ;OVERSIZE IN Z
   Q505=+0.2  ;FINISHING FEED RATE
   Q463=+2    ;MAX. CUTTING DEPTH
   Q507=+0    ;MACHINING DIRECTION
   Q508=+0    ;OFFSET WIDTH
   Q509=+0    ;DEPTH COMPENSATION
   Q488=+0    ;PLUNGING FEED RATE
12 L  X+65 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

13.20 AXIAL RECESSING EXTENDED (Cycle 852, ISO: G852)

Application



This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter Q491 is less than the end diameter Q493, the cycle runs inside machining.

Roughing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

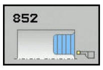
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

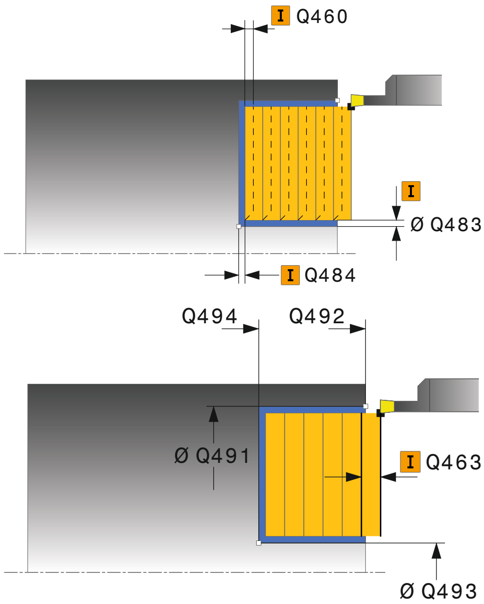
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the parallel line to the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the parallel line to the rotary axis
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)



Example

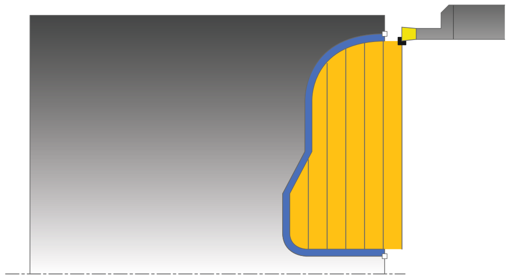
11 CYCL DEF 852 ENH.REC.TURNING, AX.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE
Q496=+5	;ANGLE OF SECOND SIDE

- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.

```
Q503=+1     ;TYPE OF END ELEMENT
Q504=+0.5   ;SIZE OF END ELEMENT
Q478=+0.3   ;ROUGHING FEED RATE
Q483=+0.4   ;OVERSIZE FOR DIAMETER
Q484=+0.2   ;OVERSIZE IN Z
Q505=+0.2   ;FINISHING FEED RATE
Q463=+2     ;MAX. CUTTING DEPTH
Q507=+0     ;MACHINING DIRECTION
Q508=+0     ;OFFSET WIDTH
Q509=+0     ;DEPTH COMPENSATION
Q488=+0     ;PLUNGING FEED RATE
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

13.21 AXIAL RECESSING (Cycle 850, ISO: G850)

Application



This cycle enables you to recess slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The control positions the tool at rapid traverse in the X coordinate (first recessing position).
2 The control performs a recessing traverse until the first plunging depth is reached.
3 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
4 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6 The tool recesses to the next plunging depth.
7 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side walls of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

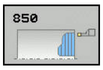
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

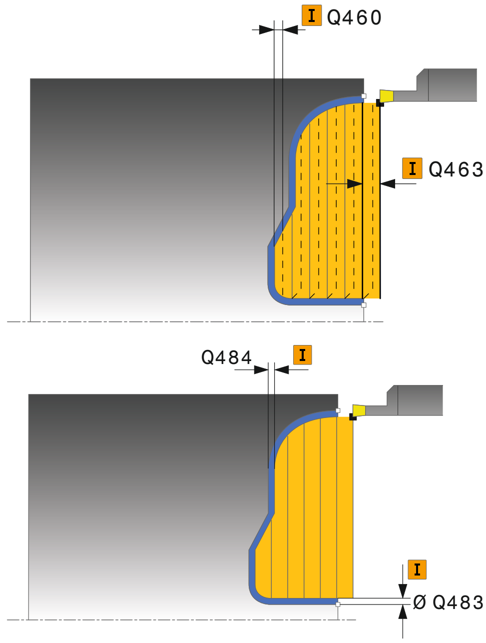
Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

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

From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).

Cycle parameters





- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q488 Feed rate for plunging (0=auto)?**: Feed rate for machining of plunging elements. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction

- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q507 Direction (0=bidir./1=unidir.)?**: Cutting direction:
  0: Bidirectional (in both directions)
  1: Unidirectional (in contour direction)
- **Q508 Offset width?**: Reduction of cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width.
- **Q509 Depth compensat. for finishing?**: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation function.
- **Q499 Reverse contour (0=no/1=yes)?**: Machining direction:
  0: Contour machined in the programmed direction
  1: in reverse direction to the programmed direction

Example

```
9  CYCL DEF 14.0  CONTOUR
10  CYCL DEF 14.1  CONTOUR LABEL2
11 CYCL DEF 850  RECESS TURNG,
   AXIAL
    Q215=+0     ;MACHINING OPERATION
    Q460=+2     ;SAFETY CLEARANCE
    Q478=+0.3   ;ROUGHING FEED RATE
    Q488=0      ;PLUNGING FEED RATE
    Q483=+0.4   ;OVERSIZE FOR DIAMETER
    Q484=+0.2   ;OVERSIZE IN Z
    Q505=+0.2   ;FINISHING FEED RATE
    Q479=+0     ;CONTOUR MACHINING
                LIMIT
    Q480=+0     ;DIAMETER LIMIT VALUE
    Q482=+0     ;LIMIT VALUE Z
    Q463=+2     ;MAX. CUTTING DEPTH
    Q507=+0     ;MACHINING DIRECTION
    Q508=+0     ;OFFSET WIDTH
    Q509=+0     ;DEPTH COMPENSATION
    Q499=+0     ;REVERSE CONTOUR
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z+0
17 L  Z-10
18 RND  R5
19 L  X+40 Z-15
20 L  Z+0
21 LBL  0
```

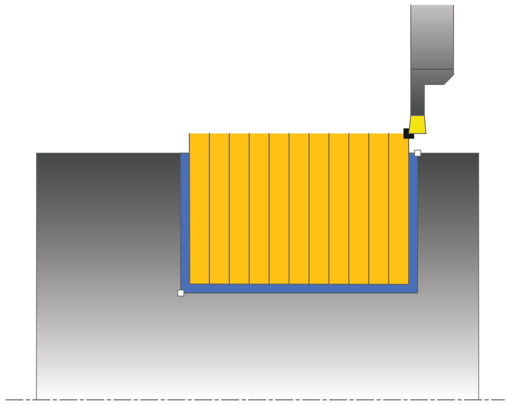
13.22 RADIAL RECESSING (Cycle 861, ISO: G861)

Application

This cycle enables you to radially cut in right-angled slots.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

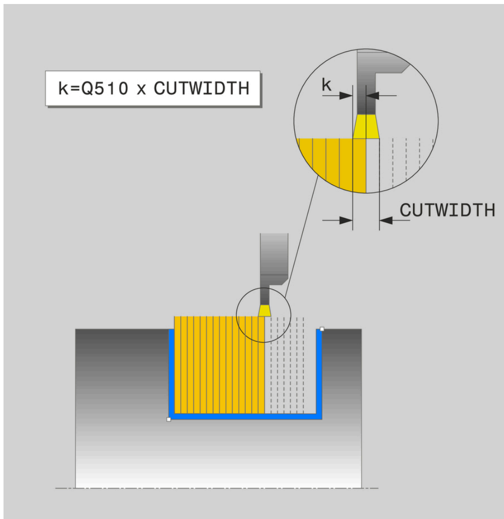
The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.



Roughing cycle run

The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.



Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

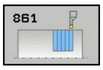
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

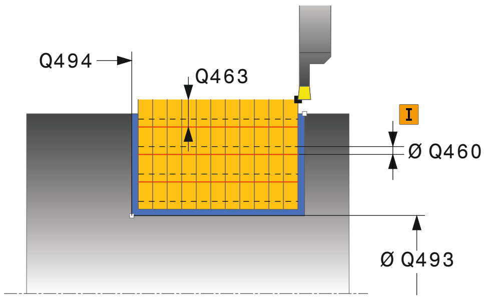
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

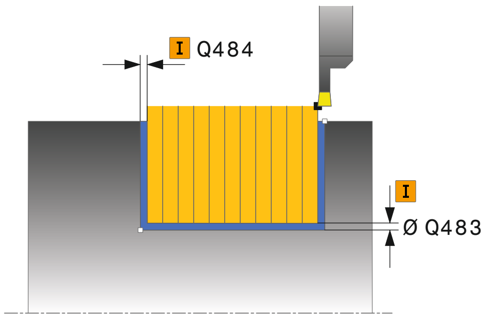
FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0,001 to 1





Example

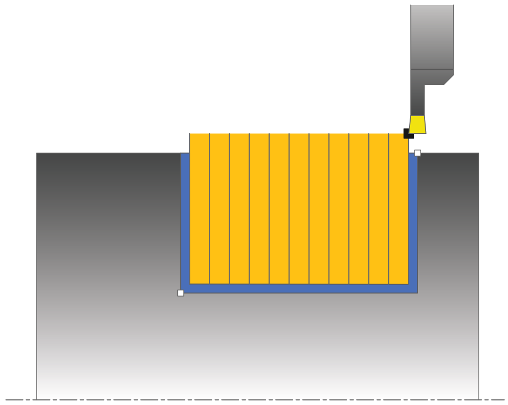
```
11 CYCL DEF 861 SIMPLE RECESS,
   RADL.
   Q215=+0     ;MACHINING OPERATION
   Q460=+2     ;SAFETY CLEARANCE
   Q493=+50    ;DIAMETER AT CONTOUR
               END
   Q494=-50    ;CONTOUR END IN Z
   Q478=+0.3   ;ROUGHING FEED RATE
   Q483=+0.4   ;OVERSIZE FOR DIAMETER
   Q484=+0.2   ;OVERSIZE IN Z
   Q505=+0.2   ;FINISHING FEED RATE
```

- **Q511 Feed rate factor in %?** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width defined in **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The control thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0,001 to 150
- **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
    Q463=+0     ;LIMIT TO DEPTH
    Q510=+0.8   ;RECESSING OVERLAP
    Q511=+100   ;FEED RATE FACTOR
    Q462=0      ;RETRACTION MODE
    Q211=3      ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z-25 FMAX  M303
13 CYCL CALL
```

13.23 RADIAL RECESSING EXTENDED (Cycle 862, ISO: G862)

Application



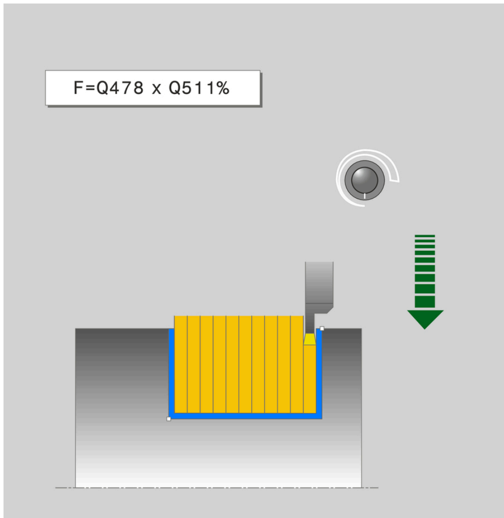
This cycle enables you to radially cut in slots. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run



1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

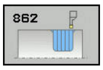
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

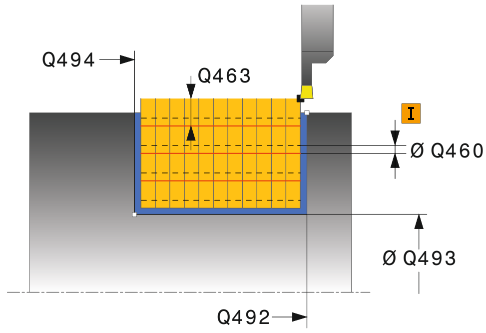
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

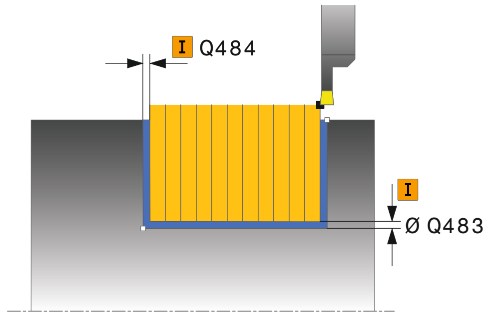
FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the perpendicular to the rotary axis.
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the perpendicular to the rotary axis.
- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)





Example

11 CYCL DEF 862 EXPND. RECESS, RADL.	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE
Q501=+1	;TYPE OF STARTING ELEMENT
Q502=+0.5	;SIZE OF STARTING ELEMENT
Q500=+1.5	;RADIUS OF CONTOUR EDGE

- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0,001 to 1
- **Q511 Feed rate factor in %?** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width defined in **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The control thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0,001 to 150
- **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
    Q496=+5      ;ANGLE OF SECOND SIDE
    Q503=+1      ;TYPE OF END ELEMENT
    Q504=+0.5    ;SIZE OF END ELEMENT
    Q478=+0.3    ;ROUGHING FEED RATE
    Q483=+0.4    ;OVERSIZE FOR DIAMETER
    Q484=+0.2    ;OVERSIZE IN Z
    Q505=+0.2    ;FINISHING FEED RATE
    Q463=+0      ;LIMIT TO DEPTH
    Q510=0.8     ;RECESSING OVERLAP
    Q511=+100    ;FEED RATE FACTOR
    Q462=+0      ;RETRACTION MODE
    Q211=3       ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

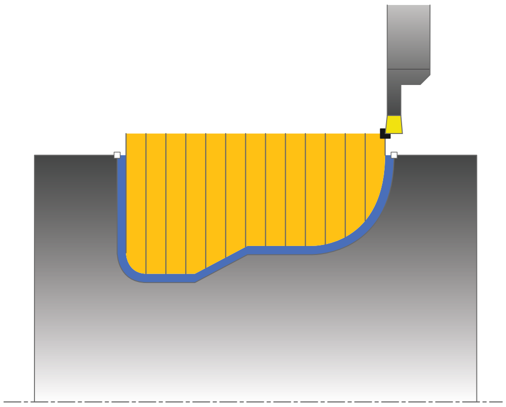
13.24 RECESSING CONTOUR RADIAL (Cycle 860, ISO: G860)

Application

This cycle enables you to radially cut in slots of any form.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.



Roughing cycle run

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

- Before the cycle is called position the tool so that it is already on the side of the cutting limit on which the material is to be cut

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

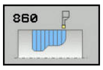
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

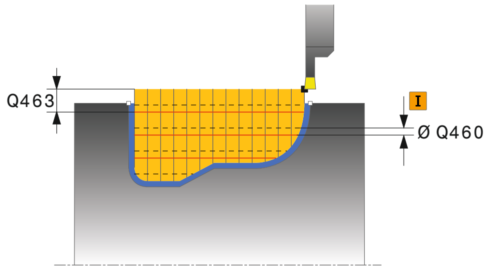
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

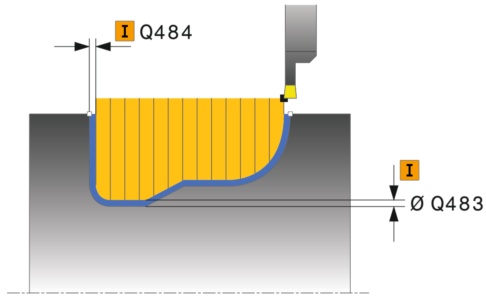
FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut





Example

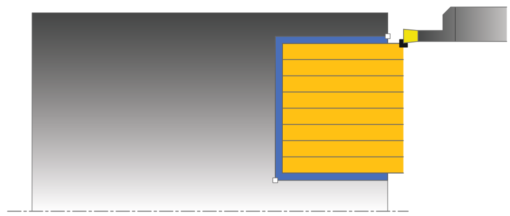
9 CYCL DEF 14.0 CONTOUR	
10 CYCL DEF 14.1 CONTOUR LABEL2	
11 CYCL DEF 860 CONT. RECESS, RADIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE

- **Q510 Overlap factor for recess width?** Factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0,001 to 1
- **Q511 Feed rate factor in %?** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width defined in **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The control thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0,001 to 150
- **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
   Q479=+0     ;CONTOUR MACHINING
               LIMIT
   Q480=+0     ;DIAMETER LIMIT VALUE
   Q482=+0     ;LIMIT VALUE Z
   Q463=+0     ;LIMIT TO DEPTH
   Q510=0.08   ;RECESSING OVERLAP
   Q511=+100   ;FEED RATE FACTOR
   Q462=+0     ;RETRACTION MODE
   Q211=3      ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z-20
17 L  X+45
18 RND  R2
19 L  X+40 Z-25
20 L  Z+0
21 LBL  0
```

13.25 AXIAL RECESSING (Cycle 871, ISO: G871)

Application



This cycle enables you to perform axial recessing of right-angled slots (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

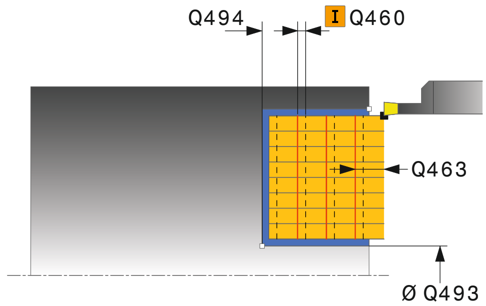
1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes half the slot width at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes half the slot width at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

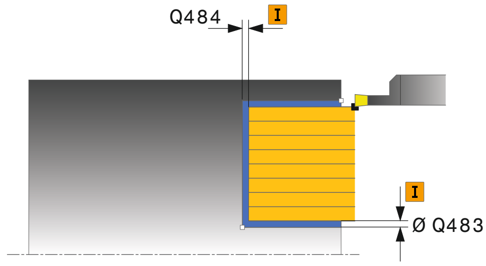
Please note while programming:

> Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
>
> The tool position at cycle call defines the size of the area to be machined (cycle starting point)
>
> **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters

- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0,001 to 1





Example

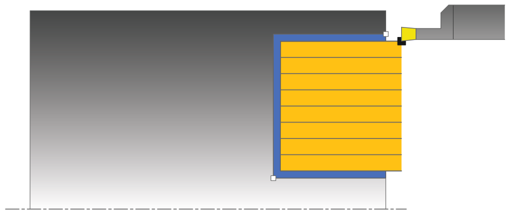
11 CYCL DEF 871 SIMPLE RECESS, AXIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-10	;CONTOUR END IN Z
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q463=+0	;LIMIT TO DEPTH

- **Q511 Feed rate factor in %?** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width defined in **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The control thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0,001 to 150
- **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
    Q510=+0.8  ;RECESSING OVERLAP
    Q511=+100  ;FEED RATE FACTOR
    Q462=0     ;RETRACTION MODE
    Q211=3     ;DWELL TIME IN REVS
12 L  X+65 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

13.26 AXIAL RECESSING EXTENDED (Cycle 872, ISO: G872)

Application



This cycle enables you to perform axial recessing of slots (face recessing). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control retracts the tool at rapid traverse.
4. The control positions the tool at rapid traverse to the second slot side.
5. The control finishes the side wall of the slot at the defined feed rate **Q505**.
6. The control finishes one half of the slot at the defined feed rate.
7. The control positions the tool at rapid traverse to the first side.
8. The control finishes the other half of the slot at the defined feed rate.
9. The control returns the tool at rapid traverse to the cycle starting point.

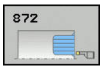
Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

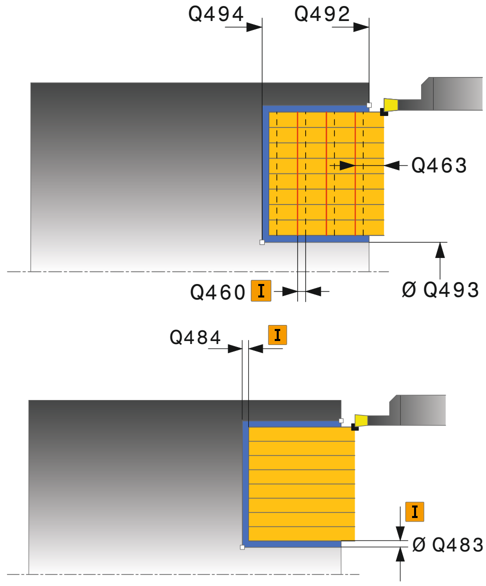
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the contour starting point
- **Q493 Diameter at end of contour?**: X coordinate of the contour end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the contour end point
- **Q495 Angle of side?**: Angle between the edge at the contour starting point and the parallel line to the rotary axis
- **Q501 Starting element type (0/1/2)?**: Define the type of element at the start of the contour (circumferential surface):
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q502 Size of starting element?**: Size of the starting element (chamfer section)
- **Q500 Radius of the contour corner?**: Radius of the inside contour corner. If no radius is specified, the radius will be that the indexable insert.
- **Q496 Angle of second side?**: Angle between the edge at the contour end point and the parallel line to the rotary axis



Example

11 CYCL DEF 871 EXPND. RECESS, AXIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q491=+75	;DIAMETER AT CONTOUR START
Q492=-20	;CONTOUR START IN Z
Q493=+50	;DIAMETER AT CONTOUR END
Q494=-50	;CONTOUR END IN Z
Q495=+5	;ANGLE OF SIDE

- **Q503 End element type (0/1/2)?**: Define the type of element at the end of the contour:
  0: No additional element
  1: Element is a chamfer
  2: Element is a radius
- **Q504 Size of end element?**: Size of the end element (chamfer section)
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0,001 to 1

```
Q501=+1    ;TYPE OF STARTING ELEMENT
Q502=+0.5  ;SIZE OF STARTING ELEMENT
Q500=+1.5  ;RADIUS OF CONTOUR EDGE
Q496=+5    ;ANGLE OF SECOND SIDE
Q503=+1    ;TYPE OF END ELEMENT
Q504=+0.5  ;SIZE OF END ELEMENT
Q478=+0.3  ;ROUGHING FEED RATE
Q483=+0.4  ;OVERSIZE FOR DIAMETER
Q484=+0.2  ;OVERSIZE IN Z
Q505=+0.2  ;FINISHING FEED RATE
Q463=+0    ;LIMIT TO DEPTH
Q510=+0.08;RECESSING OVERLAP
Q511=+100  ;FEED RATE FACTOR
Q462=0     ;RETRACTION MODE
Q211=3     ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

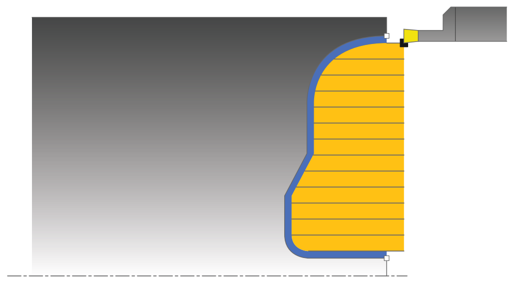
- **Q511 Feed rate factor in %?** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width defined in **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The control thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0,001 to 150
- **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

13.27 AXIAL RECESSING (Cycle 870, ISO: G870)

Application

This cycle enables you to perform axial recessing of slots of any form (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.



Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can exceed the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC 640 machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

- Before the cycle is called position the tool so that it is already on the side of the cutting limit on which the material is to be cut

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

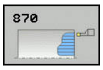
The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

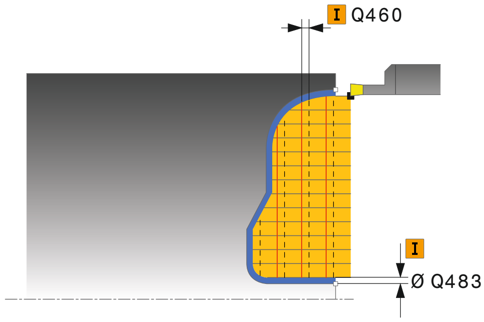
FUNCTION TURNDATA CORR TCS: Z/X DCW and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.

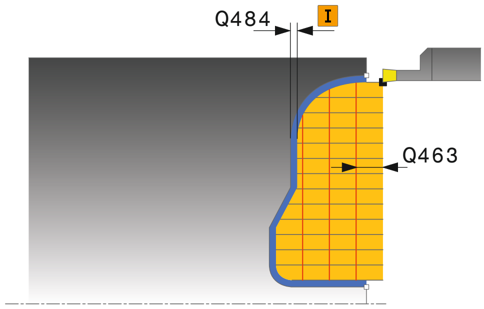
If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q460 Set-up clearance?**: Reserved, currently without function.
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999
- **Q484 Oversize in Z?** (incremental): Oversize for the defined contour in axial direction
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q479 Machining limits (0/1)?**: Activate cutting limit:
  0: No cutting limit active
  1: Cutting limit (**Q480/Q482**)
- **Q480 Value of diameter limit?**: X value for contour limitation (diameter value)
- **Q482 Value of cutting limit in Z?**: Z value for contour limitation
- **Q463 Limit to plunging depth?**: Max. recessing depth per cut
- **Q510 Overlap factor for recess width?** Factor Q510 influences the stepover of the tool when roughing. Q510 is multiplied by the **CUTWIDTH** of the tool. This results in the stepover factor "k". Input range: 0,001 to 1





Example

9 CYCL DEF 14.0 CONTOUR	
10 CYCL DEF 14.1 CONTOUR LABEL2	
11 CYCL DEF 870 CONT. RECESS, AXIAL	
Q215=+0	;MACHINING OPERATION
Q460=+2	;SAFETY CLEARANCE
Q478=+0.3	;ROUGHING FEED RATE
Q483=+0.4	;OVERSIZE FOR DIAMETER
Q484=+0.2	;OVERSIZE IN Z
Q505=+0.2	;FINISHING FEED RATE
Q479=+0	;CONTOUR MACHINING LIMIT
Q480=+0	;DIAMETER LIMIT VALUE
Q482=+0	;LIMIT VALUE Z

- **Q511 Feed rate factor in %?** Factor Q511 influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width defined in **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. That way, you can define the roughing feed rate Q478 so high that it permits optimum cutting conditions for each overlap of the cutting width (Q510). The control thus reduces the feed rate by the factor Q511 only when recessing with full contact. In total, this can lead to reduced machining times. Input range: 0,001 to 150
- **Q462 Retraction behavior (0/1)?** Q462 defines the retraction behavior after recessing.
  0: The control retracts the tool along the contour
  1: The control first moves the tool away from the contour diagonally and then retracts it
- **Q211 Dwell time / 1/min?** A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input range: 0 to 999.9999

```
    Q463=+0     ;LIMIT TO DEPTH
    Q510=0.8    ;RECESSING OVERLAP
    Q511=+100   ;FEED RATE FACTOR
    Q462=+0     ;RETRACTION MODE
    Q211=3      ;DWELL TIME IN REVS
12 L  X+75 Y+0  Z+2 FMAX  M303
13 CYCL CALL
14 M30
15 LBL  2
16 L  X+60 Z+0
17 L  Z-10
18 RND  R5
19 L  X+40 Z-15
20 L  Z+0
21 LBL  0
```

13.28 THREAD LONGITUDINAL (Cycle 831, ISO: G831)

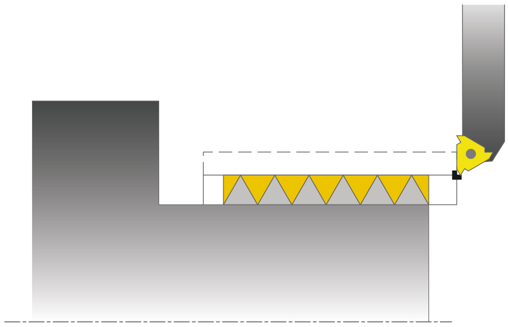
Application

This cycle enables you to run longitudinal turning of threads.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth, the cycle uses thread depth in accordance with the ISO1502 standard.

The cycle can be used for inside and outside machining.



Cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a paraxial longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The number of threads for thread cutting is limited to 500.

The control uses the set-up clearance **Q460** as approach length. The approach path must be long enough for the feed axes to be accelerated to the required velocity.

The control uses the thread pitch as idle travel path. The idle travel distance must be long enough to decelerate the feed axes.

Parameters are available for approach and overrun in Cycle 832 THREAD EXTENDED.

When the control cuts a thread, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

NOTICE

Danger of collision!

With pre-positioning in the negative diameter range, the action of parameter Q471 Thread position is reversed. In that case outside thread is 1 and inside thread is 0. Collision may occur between the tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g. to machine internal and external threads with only one tool. If with such a machine you wish to use an outside machine tool for inside machining, you can execute machining in the negative X- diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

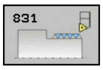
NOTICE

Danger of collision!

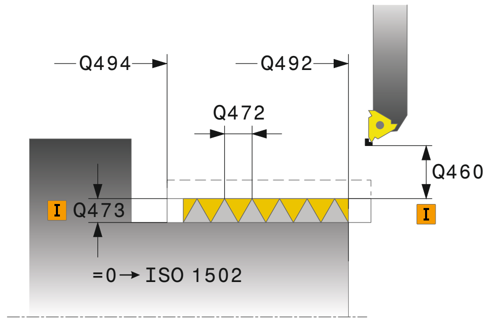
If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

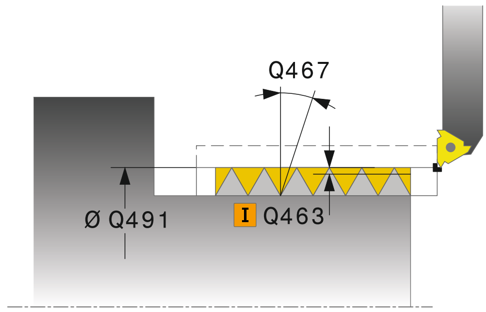
- Program the **Q467** angle of infeed to be less than the side angle of the thread.

Cycle parameters



- **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Q460 Setup clearance?**: Set-up clearance in radial and axial direction. In axial direction, the set-up clearance is used for acceleration (approach path) until the synchronized feed rate is reached.
- **Q491 Thread diameter?**: Define the nominal diameter of the thread.
- **Q472 Thread pitch?**: Pitch of the thread
- **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Q492 Contour start in Z?**: Z coordinate of the starting point
- **Q494 Contour end in Z?**: Z coordinate of the end point including the runout of the thread Q474.
- **Q474 Length of thread runout?** (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.
- **Q463 Maximum cutting depth?**: Maximum plunging depth in radial direction relative to the radius.
- **Q467 Feed angle?**: Angle for the infeed Q463. The reference angle is formed by the perpendicular to the rotary axis.
- **Q468 Infeed type (0/1)?**: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Q470 Starting angle?**: Angle of the turning spindle at which the thread should start.
- **Q475 Number of thread grooves?**: Number of thread grooves
- **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth





Example

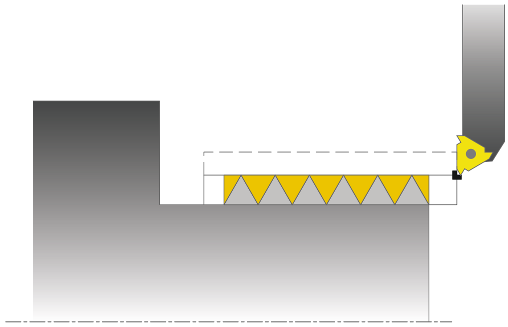
```
11 CYCL DEF 831 THREAD
   LONGITUDINAL
   Q471=+0      ;THREAD POSITION
   Q460=+5      ;SAFETY CLEARANCE
   Q491=+75     ;THREAD DIAMETER
   Q472=+2      ;THREAD PITCH
   Q473=+0      ;DEPTH OF THREAD
   Q492=+0      ;CONTOUR START IN Z
   Q494=-15     ;CONTOUR END IN Z
   Q474=+0      ;THREAD RUN-OUT
   Q463=+0.5    ;MAX. CUTTING DEPTH
   Q467=+30     ;ANGLE OF INFEED
   Q468=+0      ;TYPE OF INFEED
   Q470=+0      ;STARTING ANGLE
   Q475=+30     ;NUMBER OF STARTS
   Q476=+30     ;NUMBER OF AIR CUTS
12 L  X+80 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

13.29 THREAD EXTENDED (Cycle 832, ISO: G832)

Application

This cycle enables you to run both face turning and longitudinal turning of threads or tapered threads. Expanded scope of function:

- Selection of longitudinal thread or face thread.
- The parameters for dimension type of taper, taper angle and contour starting point X enable the definition of various tapered threads.
- The parameters for the approach length and the idle travel distance define a path in which feed axes can be accelerated and decelerated.



You can process single threads or multi-threads with the cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.

The overrun path (**Q466**) must be long enough to decelerate the feed axes.

When the control cuts a thread, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

NOTICE

Danger of collision!

With pre-positioning in the negative diameter range, the action of parameter Q471 Thread position is reversed. In that case outside thread is 1 and inside thread is 0. Collision may occur between the tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g. to machine internal and external threads with only one tool. If with such a machine you wish to use an outside machine tool for inside machining, you can execute machining in the negative X- diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

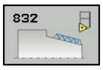
NOTICE

Danger of collision!

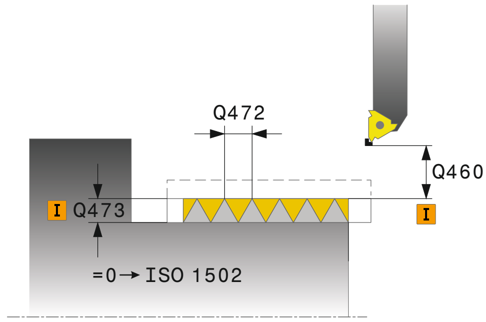
If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

- Program the **Q467** angle of infeed to be less than the side angle of the thread.

Cycle parameters



- **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Q461 Thread orientation (0/1)?**: Define the direction of the thread pitch:
  0: Longitudinal (parallel to the rotary axis)
  1: Lateral (perpendicular to the rotary axis)
- **Q460 Set-up clearance?**: Set-up clearance perpendicular to the thread pitch.
- **Q472 Thread pitch?**: Pitch of the thread
- **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Q464 Dimens. type taper (0-4)?**: Define the type of dimension for the taper contour:
  0: Via starting point and end point
  1: Via end point, start-X and taper angle
  2: Via end point, start-Z and taper angle
  3: Via starting point, end-X and taper angle
  4: Via starting point, end-Z and taper angle
- **Q491 Diameter at contour start?**: X coordinate of the contour starting point (diameter value)
- **Q492 Contour start in Z?**: Z coordinate of the starting point
- **Q493 Diameter at end of contour?**: X coordinate of the end point (diameter value)
- **Q494 Contour end in Z?**: Z coordinate of the end point
- **Q469 Taper angle (diameter)?** Taper angle of the contour
- **Q474 Length of thread runout?** (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.



Example

11 CYCL DEF 832 THREAD EXTENDED		
	Q471=+0	;THREAD POSITION
	Q461=+0	;THREAD ORIENTATION
	Q460=+2	;SAFETY CLEARANCE
	Q472=+2	;THREAD PITCH
	Q473=+0	;DEPTH OF THREAD
	Q464=+0	;DIMENSION TYPE TAPER
	Q491=+100	;DIAMETER AT CONTOUR START
	Q492=+0	;CONTOUR START IN Z
	Q493=+110	;DIAMETER AT CONTOUR END
	Q494=-35	;CONTOUR END IN Z
	Q469=+0	;TAPER ANGLE
	Q474=+0	;THREAD RUN-OUT
	Q465=+4	;STARTING PATH
	Q466=+4	;OVERRUN PATH
	Q463=+0.5	;MAX. CUTTING DEPTH
	Q467=+30	;ANGLE OF INFEED

- **Q465 Starting path?** (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside the defined thread contour.
- **Q466 Overrun path?**: Length of the path in pitch direction on which the feed axes are decelerated. The idle travel path is within the defined thread contour.
- **Q463 Maximum cutting depth?**: Maximum plunging depth perpendicular to the thread pitch
- **Q467 Feed angle?**: Angle for the infeed Q463. The reference angle is formed by a line that is parallel to the thread pitch
- **Q468 Infeed type (0/1)?**: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Q470 Starting angle?**: Angle of the turning spindle at which the thread should start.
- **Q475 Number of thread grooves?**: Number of thread grooves
- **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth

```
    Q468=+0     ;TYPE OF INFEED
    Q470=+0     ;STARTING ANGLE
    Q475=+30    ;NUMBER OF STARTS
    Q476=+30    ;NUMBER OF AIR CUTS
12 L  X+80 Y+0  Z+2 FMAX  M303
13 CYCL CALL
```

13.30 CONTOUR-PARALLEL THREAD (Cycle 830, ISO: G830)

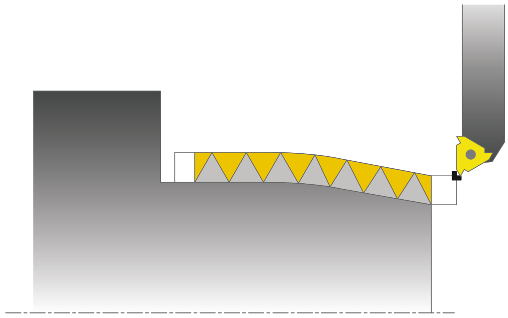
Application

This cycle enables you to run both face turning and longitudinal turning of threads with any form.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.



NOTICE

Danger of collision!

Cycle 830 runs the overrun **Q466** following the programmed contour. Take the available space into account.

- Clamp the workpiece in such a way that there is no danger of collision if the control extends the contour by Q466, Q467.

Cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control runs a thread cut parallel to the defined thread contour. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming:

Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.

The overrun path (**Q466**) must be long enough to decelerate the feed axes.

Both the approach and overrun take place outside the defined contour.

When the control cuts a thread, the feed-rate override knob is disabled. The spindle speed override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

Before the cycle call, you must program Cycle **14 CONTOUR** or **SEL CONTOUR** in order to be able to jump to the corresponding subprogram (by indicating its number).

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

NOTICE

Danger of collision!

With pre-positioning in the negative diameter range, the action of parameter Q471 Thread position is reversed. In that case outside thread is 1 and inside thread is 0. Collision may occur between the tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g. to machine internal and external threads with only one tool. If with such a machine you wish to use an outside machine tool for inside machining, you can execute machining in the negative X- diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

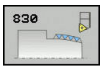
NOTICE

Danger of collision!

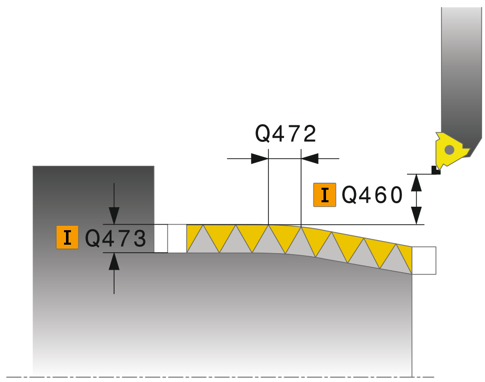
If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

- Program the **Q467** angle of infeed to be less than the side angle of the thread.

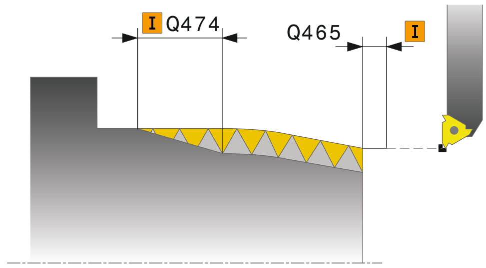
Cycle parameters



- **Q471 Thread position (0=ext./1=int.)?**: Define the position of the thread:
  0: External thread
  1: Internal thread
- **Q461 Thread orientation (0/1)?**: Define the direction of the thread pitch:
  0: Longitudinal (parallel to the rotary axis)
  1: Lateral (perpendicular to the rotary axis)
- **Q460 Set-up clearance?**: Set-up clearance perpendicular to the thread pitch.
- **Q472 Thread pitch?**: Pitch of the thread
- **Q473 Thread depth (radius)?** (incremental): Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch.
- **Q474 Length of thread runout?** (incremental): Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter Q460.



- **Q465 Starting path?** (incremental): Length of the path in pitch direction on which the feed axes are accelerated to the required velocity. The approach path is outside the defined thread contour.
- **Q466 Overrun path?**: Length of the path in pitch direction on which the feed axes are decelerated. The idle travel path is within the defined thread contour.
- **Q463 Maximum cutting depth?**: Maximum plunging depth perpendicular to the thread pitch
- **Q467 Feed angle?**: Angle for the infeed Q463. The reference angle is formed by a line that is parallel to the thread pitch
- **Q468 Infeed type (0/1)?**: Define the type of infeed:
  0: Constant chip cross section (infeed lessens with depth)
  1: Constant plunging depth
- **Q470 Starting angle?**: Angle of the turning spindle at which the thread should start.
- **Q475 Number of thread grooves?**: Number of thread grooves
- **Q476 Number of air cuts?**: Number of air cuts without infeed at finished thread depth

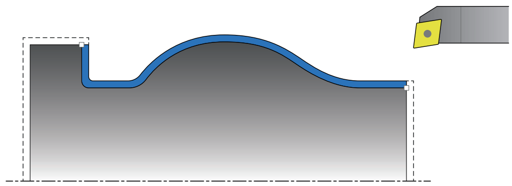


Example

9 CYCL DEF 14.0 CONTOUR
10 CYCL DEF 14.1 CONTOUR LABEL2
11 CYCL DEF 830 THREAD CONTOUR-PARALLEL
Q471=+0 ;THREAD POSITION
Q461=+0 ;THREAD ORIENTATION
Q460=+2 ;SAFETY CLEARANCE
Q472=+2 ;THREAD PITCH
Q473=+0 ;DEPTH OF THREAD
Q474=+0 ;THREAD RUN-OUT
Q465=+4 ;STARTING PATH
Q466=+4 ;OVERRUN PATH
Q463=+0.5 ;MAX. CUTTING DEPTH
Q467=+30 ;ANGLE OF INFEED
Q468=+0 ;TYPE OF INFEED
Q470=+0 ;STARTING ANGLE
Q475=+30 ;NUMBER OF STARTS
Q476=+30 ;NUMBER OF AIR CUTS
12 L X+80 Y+0 Z+2 FMAX M303
13 CYCL CALL
14 M30
15 LBL 2
16 L X+60 Z+0
17 L X+70 Z-30
18 RND R60
19 L Z-45
20 LBL 0

13.31 TURNING SIMULTANEOUS FINISHING (Cycle 883, ISO: G883, software option #158)

Application



You can use this cycle to machine complex contours that are only accessible with different inclinations. When machining with this cycle, the inclination between tool and workpiece changes. This results in machining operations with at least 3 axes (two linear axes and one rotary axis).

The cycle monitors the workpiece contour with respect to the tool and the tool carrier. The cycle avoids unnecessary tilting movements in order to machine optimum surfaces.
If you want to force tilting movements, you can define inclination angles at the beginning and at the end of the contour. Even if simple contours have to be machined, you can use a large area of the indexable insert to achieve longer tool life.

You define the contour in a subprogram and reference it using Cycle 14 or SEL CONTOUR.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The tool retracts to set-up clearance Q460. The movement is performed at rapid traverse.
2 If programmed, the tool traverses to the inclination angle that was calculated by the control based on the minimum and maximum inclination angles you have defined.
3 The control finishes the contour of the finished part (contour starting point to contour end point) simultaneously at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

Please note while programming!

Cycle 883 **TURNING SIMULTANEOUS FINISHING** is machine-dependent. Refer to your machine manual!

If the tilting axis is not perpendicular to the axis of the turning spindle, an error message will be displayed.

Based on the programmed parameters, the control calculates only **one** collision-free path.

Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Before calling the cycle, you need to program **FUNCTION TCPM** with the **REFPNT TIP-CENTER** tool center point.

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

The inclination angles **Q556** and **Q557** are limited by software limit switches.

If you deactivate monitoring by the software limit switches in the Test Run operating mode, the path might be different from the one that will actually be machined.

Please note: The smaller the resolution in cycle parameter **Q555** is, the easier will it be to find a solution even in complex situations. The drawback is that the calculation will take more time.

The cycle requires the tool holder geometry. Define it in the KINEMATIC column of the tool table (tool.t). The cycle monitors a 2-D cut with respect to the workpiece contour. The depth of the holder will **not** be monitored

Please note that cycle parameters **Q565** (Finishing allowance in diameter) and **Q566** (Finishing allowance in Z) cannot be combined with **Q567** (Finishing allowance of contour)!

NOTICE

Danger of collision!

The control does not monitor any collisions between the tool and the workpiece (DCM). Incorrect pre-positioning can also lead to contour damage. Danger of collision during machining!

- Program a suitable pre-position
- Check the machining sequence and the contour with the aid of the graphic simulation as well as by running the program slowly in Program Run, Full Sequence operating mode.

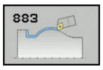
NOTICE

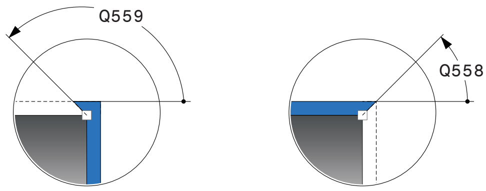
Danger of collision!

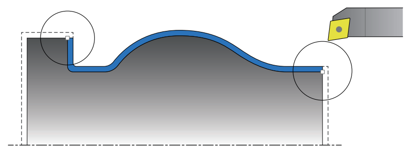
If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining.

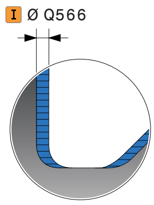
- Make sure to clamp the workpiece in such a way that it projects far enough from the fixture and no collision can occur between tool and fixture.

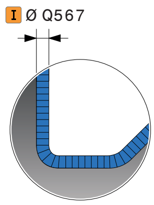
Cycle parameters

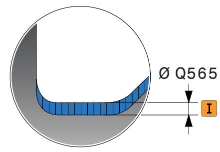












- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioning
- **Q499 Reverse the contour (0-2)?**: Define the machining direction of the contour:
  0: Contour machined in the programmed direction
  1: Contour machined in reverse direction to the programmed direction
  2: Contour machined in reverse direction to the programmed direction; additionally, the orientation of the tool is adjusted
- **Q558 Extensn. angle at contour start?**: The contour is extended by a line from the contour starting point with the angle indicated here. The control tries to approach the extension tangentially (WPL-CS)
- **Q559 Extension angle at contour end?**: The contour is extended by a line from the contour end point with the angle indicated here. The control tries to depart from the extension tangentially (WPL-CS)
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q556 Minimum angle of inclination?**: Minimum permitted angle of incidence between the tool (Z direction of the tool) and the workpiece (Z direction of the turning spindle)
- **Q557 Maximum angle of inclination?**: Maximum permitted angle of incidence between the tool (Z direction of the tool) and the workpiece (Z direction of the turning spindle)

- **Q555 Stepping angle for calculation?**: Increment for the calculation of possible solutions. Input range: 0.5 to 9.99
- **Q537 Inclin. angle (0=N/1=J/2=S/3=E)?**: Specify whether inclination angle are active:
  0: No inclination angles active
  1: Inclination angle active
  2: Inclination angle at contour start is active
  3: Inclination angle at the contour end is active
- **Q538 Inclin. angle at contour start?**: Inclination angle at the beginning of the programmed contour (WPL CS)
- **Q539 Inclinatn. angle at contour end?**: Inclination angle at the beginning of the programmed contour (WPL-CS)
- **Q565 Finishing allowance in diameter** (inkremental): Diameter allowance to remain on the contour after finishing.
- **Q566 Finishing allowance in Z?** (incremental): Allowance for the programmed contour in axial direction; material to remain on the contour after finishing.
- **Q567 Finishing allowance of contour?** (incremental): Contour-parallel oversize for the defined contour; material to remain on the contour after finishing.

Example

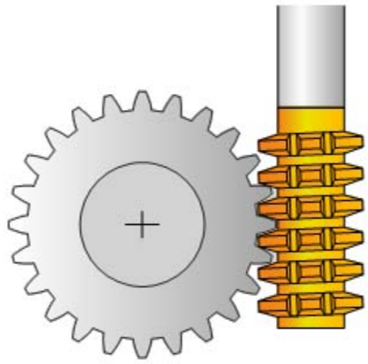
```
11 CYCL DEF 883  TURNING
   SIMULTANEOUS FINISHING
   Q460=+2    ;SETUP CLEARANCE?
   Q499=+0    ;REVERSE CONTOUR
   Q558=+0    ;EXT:ANGLE CONT.START
   Q559=+90   ;CONTOUR END EXT ANGL
   Q505=+0.2  ;FINISHING FEED RATE
   Q556=-30   ;MIN. INCLINAT. ANGLE
   Q557=+30   ;MAX. INCLINAT. ANGLE
   Q555=+7    ;STEPPING ANGLE
   Q537=+0    ;INCID. ANGLE ACTIVE
   Q538=+0    ;INCLIN. ANGLE START
   Q539=+0    ;INCLINATN. ANGLE END
   Q565=+0    ;FINISHING ALLOW. D.
   Q566=+0    ;FINISHING ALLOW. Z
   Q567=+0    ;FINISH. ALLOW. CONT.
12 L  X+58 Y+0 FMAX  M303
13 L Z+50 FMAX
14 CYCL CALL
```

13.32 GEAR HOBBING (Cycle 880, ISO: G880)

Cycle run

With Cycle 880 Gear Hobbing you can machine external cylindrical gears or helical gears with any angles. In the cycle you first define the **gear** and then the **tool** with which the gear is to be machined. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary motion of the tool spindle and rotary table. In addition, the gear hob moves along the workpiece in axial direction.

While Cycle 880 Gear Hobbing is active, the coordinate system might be rotated. It is therefore essential to program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145** at the end of the cycle.



Cycle run:

1 The control positions the tool in the tool axis to clearance height Q260 at the feed rate FMAX. If the tool is already at a location in the tool axis higher than Q260, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the FMAX feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control then tilts the working plane at the feed rate Q253; **M144** is internally active in the cycle.
4 The control positions the tool at the feed rate FMAX to the starting point in the working plane.
5 The control then moves the tool in the tool axis at the feed rate Q253 to the set-up clearance Q460.
6 The control moves the tool at the defined feed rate Q478 (for roughing) or Q505 (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z Q551+Q460 and the end point in Z Q552+Q460.
7 When the tool reaches the end point, it is retracted at the feed rate Q253 and returns to the starting point.
8 The control repeats the steps 5 to 7 until the defined gear is completed.
9 Finally, the control retracts the tool to the clearance height Q260 at the feed rate FMAX.
10 The machining operation ends in the tilted system.
11 Now you need to move the tool to a safe height and reset the tilting of the working plane.
12 Then you must program Cycle 801 RESET ROTARY COORDINATE SYSTEM and **M145**.

Please note while programming:

The values entered for module, number of teeth and outside diameter are monitored. If these values are not coherent, an error message displays. You can fill in 2 of the 3 parameters. Enter 0 for the module, the number of teeth, or the outside diameter. In this case, the control will calculate the missing value.

Program FUNCTION TURNDATA SPIN VCONST:OFF.

If you program FUNCTION TURNDATA SPIN VCONST:OFF S15, the spindle speed of the tool is calculated as follows: Q541 x S. With Q541=238 and S=15, this would result in a tool spindle speed of 3570 rpm.

Define the tool as a milling cutter in the tool table.

In order to avoid that the maximum permissible spindle speed of the tool is not exceeded, you can program a limitation. (Specify it in the **Nmax** column of the "tool.t" tool table.)

Before starting the cycle, program the direction of rotation of the workpiece (M303/M304).

Before cycle call, set the preset to the center of rotation.

Cycle 880 Gear Hobbing is run in turning mode and is CALL-active.

Software option 50 must be enabled.

Software option 131 must be enabled.

NOTICE

Danger of collision!

If you do not position the tool to a safe position, a collision may occur between the tool and workpiece (fixtures) during tilting.

- Pre-position the tool so that it is already on the desired machining side Q550.
- Move the tool to a safe position on this machining side

NOTICE

Danger of collision!

If you clamp the workpiece too tightly to the fixtures, a collision may occur between the tool and fixtures during execution. The starting point in Z and the end point in Z are extended by the set-up clearance Q460!

- Clamp the workpiece out of the fixtures far enough to prevent a danger of collision between the tool and the fixtures
- Clamp your component out of the fixtures far enough so that the extension automatically approached by the cycle of the starting point and end point by the set-up clearance Q460 does not cause a collision

NOTICE

Danger of collision!

Depending on whether you use M136 or not, the feed rate values will be interpreted differently by the control. If the programmed feed rate was too high, the workpiece might be damaged.

- If you program M136 explicitly before the cycle, the control will interpret the feed rates in the cycle in mm/rev.
- If you do not program M136 before the cycle, the control will interpret the feed rates in the cycle in mm/min.

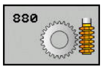
NOTICE

Danger of collision!

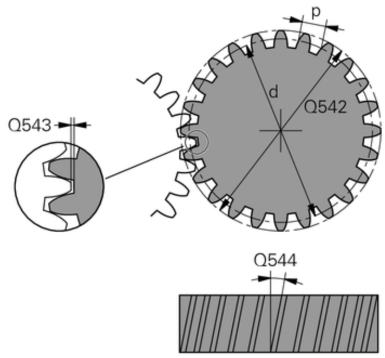
If you do not reset the coordinate system after Cycle 880, the precession angle set by the cycle will remain in effect.

- Make sure to program Cycle 801 after Cycle 800 in order to reset the coordinate system.
- Make sure to program Cycle 801 after a program abort in order to reset the coordinate system.

Cycle parameters



- **Q215 Machining operation (0/1/2/3)?**: Define machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing to finished dimension
  3: Only finishing to oversize
- **Q540 Module?**: Define the gear: Module of the gear wheel. Input range 0 to 99.9999
- **Q541 Number of teeth?**: Define the gear: Number of teeth. Input range 0 to 99999
- **Q542 Outside diameter?**: Define the gear: Outside diameter of the finished part. Input range 0 to 99999.9999
- **Q543 Trough-to-tip clearance?**: Define the gear: Distance between the tip circle of the gear to be cut and the root circle of the mating gear. Input range 0 to 9.9999
- **Q544 Angle of inclination?**: Define the gear: Angle by which teeth of a helical gear helical teeth are inclined relative to the direction of the axis (For straight-cut gears this angle is 0°.) Input range: -60 to +60
- **Q545 Tool lead angle?**: Define the tool: Angle of the tooth sides of the gear hob. Enter this value in decimal notation. (Example: 0°47'=0.7833) Input range: -60.0000 to +60.0000
- **Q546 Reverse tool rotation direction?**: Define the tool: Direction of spindle rotation of the gear hob:
  3: Tool turns clockwise (M3)
  4: Tool turns anti-clockwise (M4)
- **Q547 Angle offset of tool spindle?**: Angle by which the control rotates the workpiece at the beginning of cycle. Input range -180.0000 to +180.0000
- **Q550 Machining side (0=pos./1=neg.)?**: Define the side on which the machining operation is to be performed.
  0: Positive machining side of the reference axis in the I-CS
  1: Negative machining side of the reference axis in the I-CS



Example

63 CYCL DEF 880 GEAR HOBBING	
Q215=0	;MACHINING OPERATION
Q540=0	;MODULE
Q541=0	;NUMBER OF TEETH
Q542=0	;OUTSIDE DIAMETER
Q543=0.167	;TROUGH-TIP CLEARANCE
Q544=0	;ANGLE OF INCLINATION
Q545=0	;TOOL LEAD ANGLE
Q546=3	;CHANGE TOOL DIRECTN.
Q547=0	;ANG. OFFSET, SPINDLE
Q550=1	;MACHINING SIDE
Q533=0	;PREFERRED DIRECTION
Q530=2	;INCLINED MACHINING
Q253=750	;F PRE-POSITIONING
Q260=100	;CLEARANCE HEIGHT
Q553=10	;TOOL LENGTH OFFSET
Q551=0	;STARTING POINT IN Z
Q552=-10	;END POINT IN Z
Q463=1	;MAX. CUTTING DEPTH
Q460=2	;SAFETY CLEARANCE
Q488=0.3	;PLUNGING FEED RATE
Q478=0.3	;ROUGHING FEED RATE
Q483=0.4	;OVERSIZE FOR DIAMETER
Q505=0.2	;FINISHING FEED RATE

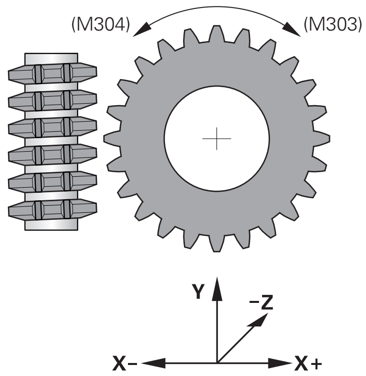
- **Q533 Preferred dir. of incid. angle?**: Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on the machine. In general, there are always two possible solutions. Using parameter Q533, configure which of the solution options the control should use:
  0: Option with the shortest distance from the current position
  -1: Option in the range between 0° and -179.9999°
  +1: Option in the range between 0° and +180°
  -2: Option in the range between -90° and -179.9999°
  +2: Option in the range between +90° and +180°
- **Q530 Inclined machining?**: Position the tilting axes for inclined machining:
  1: Position the tilting axis automatically and orient the tool tip (MOVE). The relative position between workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the tool tip (TURN)
- **Q253Feed rate for pre-positioning?**: Traversing speed of the tool when tilting and pre-positioning, and when positioning the tool axis between the individual infeeds. Input in mm/min. Input range 0 to 99999.9999 alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q553 TOOL:L offset, machining start?** (incremental): Define the length offset (L OFFSET) from which on the tool will machine. The tool will be offset in the longitudinal direction by this value. Input range: 0 to 999.9999

- **Q551 Starting point in Z?**: Starting point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- **Q552 End point in Z?**: End point in Z for gear hobbing. Input range -99999.9999 to 99999.9999
- **Q463 Maximum cutting depth?**: Maximum infeed (radius value) in radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input range 0.001 to 999.999
- **Q460 Set-up clearance?** (incremental): Distance for retraction and pre-positioningInput range: 0 to 999.999
- **Q488 Feed rate for plunging**: Feed rate for tool infeed. Input range 0 to 99999.999
- **Q478 Roughing feed rate?**: Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.
- **Q483 Oversize for diameter?** (incremental): Diameter oversize for the defined contour. Input range: 0 to 99.999.
- **Q505 Finishing feed rate?**: Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136 in millimeters per minute.

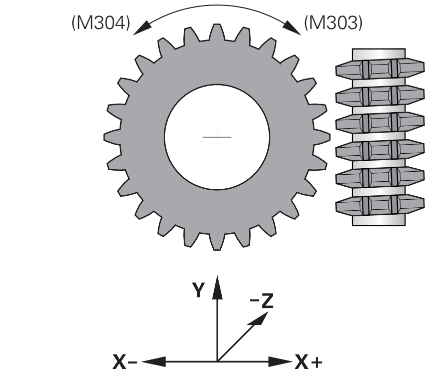
Direction of rotation depending on the machining side (Q550)

Determine the direction of rotation of the rotary table:

1 **What tool? (Right-cutting/left-cutting?)**
2 **What machining side? X+ (Q550=0) / X- (Q550=1)**
3 **Look up the direction of rotation of the rotary table in one of the two tables below!** To do so, select the appropriate table for the direction of rotation of your tool (**right-cutting/left-cutting**). Please refer to the tables below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)** ab.



Tool: Right-cutting M3	
Machining side X+ (Q550=0)	Direction of rotation of the table: Clockwise (M303)
Machining side X- (Q550=1)	Direction of rotation of the table: Counterclockwise (M304)

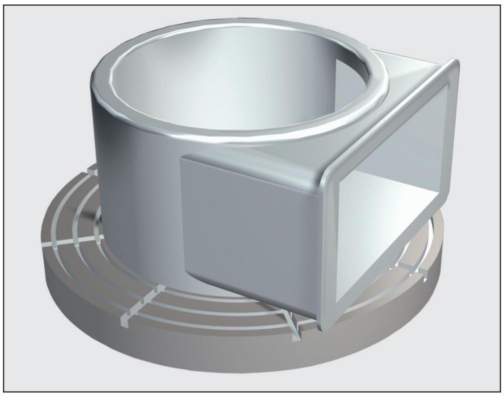


Tool: Left-cutting M4	
Machining side X+ (Q550=0)	Direction of rotation of the table: Counterclockwise (M304)
Machining side X- (Q550=1)	Direction of rotation of the table: Clockwise (M303)

13.33 CHECK UNBALANCE (Cycle 892, ISO: G892)

Application

An unbalance can occur when turning an unsymmetrical workpiece, such as a pump body. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle **892 CHECK IMBALANCE** the control checks the unbalance of the turning spindle. This cycle uses two parameters. Q450 describes the maximum unbalance and Q451 the maximum spindle speed. **If the maximum unbalance is exceeded, an error message is displayed and the NC program is aborted.** If the maximum unbalance is not exceeded, the control executes the NC program without interruption. This function protects the machine mechanics. It enables you to take action if an excessive unbalance is detected.



Please note while programming!

NOTICE

Danger of collision!

Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any imbalance. If a large unbalance is not compensated, this can lead to defects on the machine.

- Before starting a new machining cycle, run Cycle 892.
- If required, use balancing weights to compensate any unbalance.

NOTICE

Danger of collision!

The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps. If high unbalance loads are not compensated, this can lead to defects on the machine.

- Make sure to run Cycle 892 between the machining steps.
- If required, use balancing weights to compensate any unbalance.

NOTICE

Danger of collision!

High unbalance loads, especially in combination with a high mass, might damage the machine. Keep in mind the mass and unbalance of the workpiece when choosing the speed.

- Do not program high speeds with heavy workpieces or high unbalance loads.

Software option 50 must be enabled.

This function is executed in turning mode. **FUNCTION MODE TURN** must be active, otherwise display an error message.

Your machine tool builder configures Cycle 892.

Your machine tool builder defines the function of Cycle 892.

The turning spindle rotates during the unbalance check.

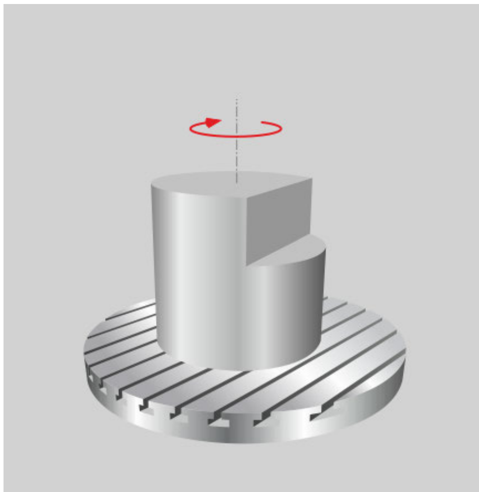
This function can also be run on machines with more than one turning spindle. Contact the machine tool builder for further information.

You need to check the applicability of the control's internal unbalance functionality for each of your machine types. If the unbalance amplitude of the turning spindle has very little effect on the adjoining axes, it might not be possible to calculate useful unbalance values from the determined results. In this case, you will have to use a system with external sensors for unbalance monitoring.

If Cycle 892 CHECK UNBALANCE has aborted the NC program, it is recommended to use the manual MEASURE UNBALANCE cycle. With this cycle, the control determines the unbalance and calculates the mass and position of a balancing weight.
Further information: User's Manual for Setup, Testing and Running NC programs

Cycle parameters

- **Q450 Max. permissible runout?** Specifies the maximum runout of a sinusoidal unbalance signal in millimeters (mm). The signal results from the following error of the measuring axis and from the spindle revolutions.
- **Q451 Rotational speed?** Enter the rotational speed in revolutions per minute. The test for an unbalance begins with a low initial speed (e.g. 50 rpm). It is then automatically increased by the specified increment (e.g. 25 rpm) until the maximum speed specified in parameter Q451 is reached. Spindle speed override is disabled.

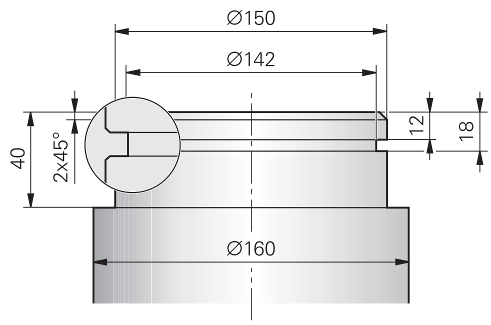


Example

```
63 CYCL DEF 892 CHECK IMBALANCE
    Q450=0      ;MAXIMUM RUNOUT
    Q451=50     ;SPEED
```

13.34 Programming example

Example: Shoulder with recess



0 BEGIN PGM SHOULDER MM	
1 BLK FORM 0.1 Y X+0 Y-10 Z-35	Workpiece blank definition
2 BLK FORM 0.2 X+87 Y+10 Z+2	
3 TOOL CALL 12	Tool call
4 M140 MB MAX	Retract the tool
5 FUNCTION MODE TURN	Activate Turning mode
6 FUNCTION TURNDATA SPIN VCONST:ON VC:150	Constant surface speed
7 CYCL DEF 800 ADJUST XZ SYSTEM	Cycle definition: adapt the coordinate system
Q497=+0 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=0 ;INCLINED MACHINING	
Q531=+0 ;ANGLE OF INCIDENCE	
Q532=750 ;FEED RATE	
Q533=+0 ;PREFERRED DIRECTION	
Q535=3 ;ECCENTRIC TURNING	
Q536=0 ;ECCENTRIC W/O STOP	
8 M136	Feed rate in mm per revolution
9 L X+165 Y+0 R0 FMAX	Move to starting point in the plane
10 L Z+2 R0 FMAX M304	Set-up clearance, turning spindle on
11 CYCL DEF 812 SHOULDER, LONG. EXT.	Cycle definition: shoulder, longitudinal
Q215=+0 ;MACHINING OPERATION	
Q460=+2 ;SAFETY CLEARANCE	
Q491=+160 ;DIAMETER AT CONTOUR START	
Q492=+0 ;CONTOUR START IN Z	
Q493=+150 ;DIAMETER AT CONTOUR END	
Q494=-40 ;CONTOUR END IN Z	
Q495=+0 ;ANGLE OF CIRCUM. SURFACE	
Q501=+1 ;TYPE OF STARTING ELEMENT	
Q502=+2 ;SIZE OF STARTING ELEMENT	
Q500=+1 ;RADIUS OF CONTOUR EDGE	

Q496=+0	;ANGLE OF FACE	
Q503=+1	;TYPE OF END ELEMENT	
Q504=+2	;SIZE OF END ELEMENT	
Q463=+2.5	;MAX. CUTTING DEPTH	
Q478=+0.25	;ROUGHING FEED RATE	
Q483=+0.4	;OVERSIZE FOR DIAMETER	
Q484=+0.2	;OVERSIZE IN Z	
Q505=+0.2	;FINISHING FEED RATE	
Q506=+0	;CONTOUR SMOOTHING	
12 CYCL CALL M8		Cycle call
13 M305		Turning spindle off
14 TOOL CALL 15		Tool call
15 M140 MB MAX		Retract the tool
16 FUNCTION TURNDATA SPIN VCONST:ON VC:100		Constant cutting speed
17 CYCL DEF 800 ADJUST XZ SYSTEM		Cycle definition: adapt the coordinate system
Q497=+0	;PRECESSION ANGLE	
Q498=+0	;REVERSE TOOL	
Q530=0	;INCLINED MACHINING	
Q531=+0	;ANGLE OF INCIDENCE	
Q532=750	;FEED RATE	
Q533=+0	;PREFERRED DIRECTION	
Q535=0	;ECCENTRIC TURNING	
Q536=+0	;ECCENTRIC W/O STOP	
18 L X+165 Y+0 R0 FMAX		Move to starting point in the plane
19 L Z+2 R0 FMAX M304		Set-up clearance, turning spindle on
20 CYCL DEF 862 EXPND. RECESS, RADL.		Cycle definition: recess
Q215=+0	;MACHINING OPERATION	
Q460=+2	;SAFETY CLEARANCE	
Q491=+150	;DIAMETER AT CONTOUR START	
Q492=-12	;CONTOUR START IN Z	
Q493=+142	;DIAMETER AT CONTOUR END	
Q494=-18	;CONTOUR END IN Z	
Q495=+0	;ANGLE OF SIDE	
Q501=+1	;TYPE OF STARTING ELEMENT	
Q502=+1	;SIZE OF STARTING ELEMENT	
Q500=+0	;RADIUS OF CONTOUR EDGE	
Q496=+0	;ANGLE OF SECOND SIDE	
Q503=+1	;TYPE OF END ELEMENT	
Q504=+1	;SIZE OF END ELEMENT	
Q478=+0.3	;ROUGHING FEED RATE	
Q483=+0.4	;OVERSIZE FOR DIAMETER	
Q484=+0.2	;OVERSIZE IN Z	
Q505=+0.15	;FINISHING FEED RATE	

Q463=+0 ;LIMIT TO DEPTH	
21 CYCL CALL M8	Cycle call
22 M305	Turning spindle off
23 M137	Feed rate in mm per minute
24 M140 MB MAX	Retract the tool
25 FUNCTION MODE MILL	Activate Milling mode
26 M30	End of program
27 END PGM SHOULDER MM	

Example: turning, simultaneous finishing

The following NC program illustrates the use of Cycle 883 **TURNING SIMULTANEOUS FINISHING**.

Program run

- Tool call:turning tool
- Start turning mode
- Move to safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGINN PGM SIMULTAN MM	
1 BLK FORM CYLINDER Z D91 L40 DIST+0.5 DI+57.5	Workpiece blank definition
2 TOOL CALL "TURN"	Tool call
3 L Z+0 R0 FMAX M91	Retract the tool
4 FUNCTION MODE TURN	Activate turning mode
5 FUNCTION TURNDATA SPIN VCONST:ON VC:200 SMAX 800	Constant cutting speed
6 CYCL DEF 800 KOORD.-SYST. ANPASSEN	Cycle definition: adapt the coordinate system
Q497 =+0 ;PRECESSION ANGLE	
Q498=+0 ;REVERSE TOOL	
Q530=+2 ;INCLINED MACHINING	
Q531=+1 ;ANGLE OF INCIDENCE	
Q532=MAX ;FEED RATE	
Q533=+1 ;PREFERRED DIRECTION	
Q535=+3 ;ECCENTRIC TURNING	
Q536=+0 ;ECCENTRIC W/O STOP	
7 M145	
8 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	Activate TCPM
9 CYCL DEF 14.0 KONTUR	Define the contour label
10 CYCL DEF 14.1 KONTURLABEL 2	
11 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING	Cycle definition: turning, simultaneous finishing
Q460=+2 ;SETUP CLEARANCE?	
Q499=+0 ;REVERSE CONTOUR	
Q558=-90 ;EXT:ANGLE CONT.START	
Q559=+90 ;CONTOUR END EXT ANGL	
Q505=+0.2 ;FINISHING FEED RATE	
Q556=-80 ;MIN. INCLINAT. ANGLE	
Q557=+60 ;MAX. INCLINAT. ANGLE	
Q555=+1 ;STEPPING ANGLE	
Q537=+0 ;INCID. ANGLE ACTIVE	
Q538=+0 ;INCLIN. ANGLE START	
Q539=+50 ;INCLINATN. ANGLE END	

Q565=+0 ;FINISHING ALLOW. D.	
Q566=+0 ;FINISHING ALLOW. Z	
Q567=+0 ;FINISH. ALLOW. CONT.	
12 L X+58 Y+0 R0 FMAX M303	Approach starting point
13 L Z+50 FMAX	Set-up clearance
14 CYCL CALL	Cycle call
15 L Z+50 FMAX	
16 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	Reset the coordinate system
17 M144	Cancel M145
18 FUNCTION MODE MILL	Activate milling mode
19 M30	End of program
20 LBL 2	
21 L X+58 Y+0 Z-1.5 RR	
22 L X+61 Z+0	
23 L X+88 Z+0	
24 L X+90 Z-1	
25 L X+90 Z-8	
26 L X+88 Z-10	
27 L X+88 Z-15	
28 L X+90 Z-17	
29 L X+90 Z-25	
30 RND R0.3	
31 L X+144 Z-25	
32 LBL 0	

Example: Gear hobbing

The following NC program illustrates the use of Cycle 880 HOBBING. This programming example illustrates the machining of a helical gear, with Module=2.1.

Program run

- Tool call: Gear hob
- Start turning mode
- Approach safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R42 L150	Workpiece blank definition: cylinder
2 FUNCTION MODE MILL	Activate milling mode
3 TOOL CALL "GEAR_HOB_D75"	Call the tool
4 FUNCTION MODE TURN	Activate turning mode
5 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	Reset the coordinate system
6 M145	Deactivate M144 if still active
7 FUNCTION TURNDATA SPIN VCONST:OFF S50	Constant surface speed OFF
8 M140 MB MAX	Retract the tool
9 L A+0 R0 FMAX	Set the rotary axis to 0
10 L X+250 Y-250 R0 FMAX	Pre-position the tool in the working plane on the side on which machining will be performed
11 Z+20 R0 FMAX	Pre-position the tool in the spindle axis
12 L M136	Feed rate in mm/rev
13 CYCL DEF 880 GEAR HOBBING	Cycle definition: hobbing
Q215=+0 ;MACHINING OPERATION	
Q540=+2.1 ;MODULE	
Q541=+0 ;NUMBER OF TEETH	
Q542=+69.3 ;OUTSIDE DIAMETER	
Q543=+0.1666 ;TROUGH-TIP CLEARANCE	
Q544=-5 ;ANGLE OF INCLINATION	
Q545=+1.6833 ;TOOL LEAD ANGLE	
Q546=+3 ;CHANGE TOOL DIRECTN.	
Q547=+0 ;ANG. OFFSET, SPINDLE	
Q550=+0 ;MACHINING SIDE	
Q533=+0 ;PREFERRED DIRECTION	
Q530=+2 ;INCLINED MACHINING	
Q253=+2000 ;F PRE-POSITIONING	
Q260=+20 ;CLEARANCE HEIGHT	
Q553=+10 ;TOOL LENGTH OFFSET	
Q551=+0 ;STARTING POINT IN Z	
Q552=-10 ;END POINT IN Z	

Q463=+1 ;MAX. CUTTING DEPTH	
Q460=2 ;SAFETY CLEARANCE	
Q488=+1 ;PLUNGING FEED RATE	
Q478=+2 ;ROUGHING FEED RATE	
Q483=+0.4 ;OVERSIZE FOR DIAMETER	
Q505=+1 ;FINISHING FEED RATE	
14 CYCL CALL M303	Call the cycle, spindle on
15 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	Reset the coordinate system
16 M145	Deactivate the active M144 in the cycle
17 FUNCTION MODE MILL	Activate milling mode
18 M140 MB MAX	Retract the tool in the tool axis
19 L A+0 C+0 R0 FMAX	Reset the rotation
20 M30	End of program
21 END PGM 5 MM	

14

Using Touch Probe Cycles

14.1 General information about touch probe cycles

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

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

Touch-probe functions are not possible in combination with the **Global Program Settings** function. If at least one settings possibility is active, the control displays an error message if a manual touch-probe function is selected or when executing an automatic touch-probe cycle.

Method of function

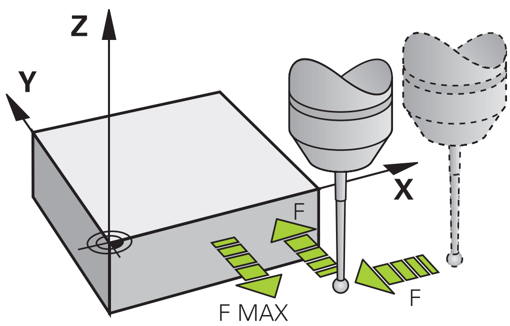
Whenever the control runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Further information: "Before You Start Working with Touch Probe Cycles", Page 537

When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the control displays an error message (distance: **DIST** from touch probe table).



Consideration of a basic rotation in the Manual Operation mode

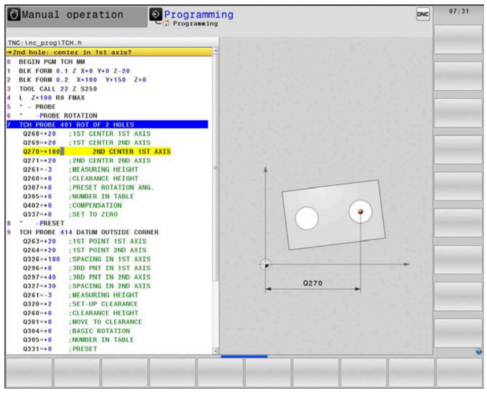
During probing, the control considers an active basic rotation and approaches the workpiece at an angle.

Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes

In the **Manual operation** and **Electronic handwheel** modes, the control provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensate workpiece misalignment
- Set presets

Touch probe cycles for automatic operation



Besides the touch probe cycles, which you can use in the Manual and El. Handwheel modes, the control provides numerous cycles for a wide variety of applications in automatic mode:

- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Presetting
- Automatic workpiece inspection
- Automatic tool measurement

You can program the touch probe cycles in the **Programming** operating mode via the **TOUCH PROBE** key. Like the most recent fixed cycles, touch probe cycles with numbers greater than 400 use Q parameters as transfer parameters. Parameters with the same function that the control requires in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the control shows a graphic during cycle definition. In the graphic, the parameter that needs to be entered is highlighted (see figure right).

Defining the touch probe cycle in Programming mode of operation

TOUCH PROBE

- The soft-key row shows all available touch probe functions divided into groups.

PRESET

- Select the desired cycle group, e.g. presetting. Cycles for automatic tool measurement are available only if your machine has been prepared for them.

410

- Select a cycle, point e.g. presetting at pocket center. The control initiates the programming dialog and asks for all required input values. At the same time, a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the control and confirm each entry with the ENT key.
- The control closes the dialog when all required data has been entered.

Soft key	Measuring cycle group	Page
ROTATION	Cycles for automatic measurement and compensation of workpiece misalignment	543
PRESET	Cycles for automatic workpiece presetting	590
MEASURING	Cycles for automatic workpiece inspection	648
SPECIAL CYCLES	Special cycles	694
CALIBRATE TS	Calibrate TS	694
KINEMATICS	Kinematics	745
TT CYCLES	Cycles for automatic tool measurement (enabled by the machine manufacturer)	784
MONITORING WITH CAMERA	Visual Setup Control (Option 136 VSC)	720

NC blocks

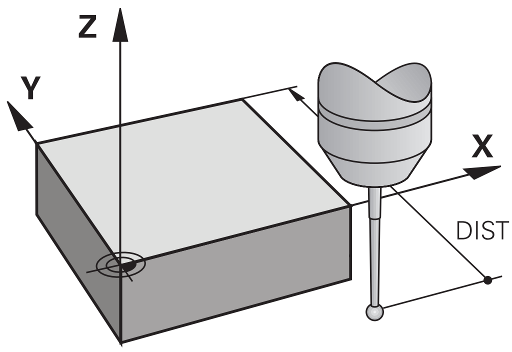
```
5 TCH PROBE 410  PRESET INSIDE
   RECTAN.
   Q321=+50   ;CENTER IN 1ST AXIS
   Q322=+50   ;CENTER IN 2ND AXIS
   Q323=60    ;FIRST SIDE LENGTH
   Q324=20    ;2ND SIDE LENGTH
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+20   ;CLEARANCE HEIGHT
   Q301=0     ;MOVE TO CLEARANCE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

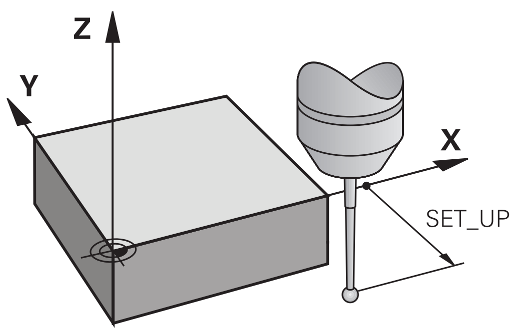
Maximum traverse to touch point: DIST in touch probe table

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



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

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



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

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

> If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch trigger probe, probing feed rate: F in touch probe table

In **F**, you define the feed rate at which the control is to probe the workpiece.

F can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).

The feed rate potentiometer may be effective with touch probe cycles. The machine tool builder defines the required settings. (the parameter **overrideForMeasure** (No. 122604) must be appropriately configured.)

Touch trigger probe, rapid traverse for positioning: FMAX

In **FMAX**, you define the feed rate at which the control pre-positions the touch probe and positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table

In **F_PREPOS**, you define whether the control is to position the touch probe at the feed rate defined in FMAX or at rapid traverse.

- Input value = **FMAX_PROBE**: Position at feed rate from **FMAX**
- Input value = **FMAX_MACHINE**: Pre-position at rapid traverse

Executing touch probe cycles

All touch probe cycles are DEF-active. This means that the control runs the cycle automatically as soon as it executes the cycle definition in the program run.

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

When running touch probe cycles, Cycle 1400 to 1499 for coordinate transformation must not be active.

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

Touch probe cycles 408 to 419 and 1400 to 1499 may be run even if a basic rotation is active. Make sure, however, that the basic rotation angle does not change when you use Cycle 7 DATUM SHIFT after the measuring cycle.

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

Touch probe cycles numbered 400 to 499 or 1400 to 1499 position the touch probe according to the following positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to clearance height and then positions it in the working plane to the first touch point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the control first positions the touch probe to the first probe point in the working plane, and then in the touch-probe axis directly to the measuring height.

14.3 Touch-probe table

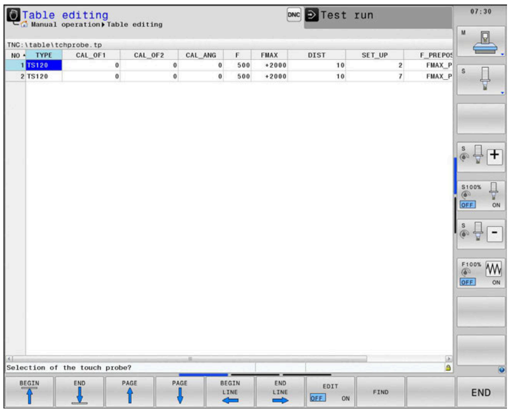
General information

Various data is stored in the touch probe table that defines the probe behavior during the probing process. If you use several touch probes on your machine tool, you can save separate data for each touch probe.

> You can also view and edit the data of the touch probe table in the expanded tool management (option 93).

Editing touch probe tables

To edit the touch probe table, proceed as follows:



- Operating mode: Press the **Manual operation** key

- Select the touch probe functions: Press the **TOUCH PROBE** soft key. The control displays additional soft keys.

- Select the touch probe table: Press the **TCH PROBE TABLE** soft key.

- Set the **EDIT** soft key to **ON**.
- Using the arrow keys, select the desired setting.
- Perform desired changes.
- Exit the touch probe table: Press the **END** soft key.

Touch probe data

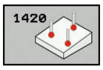
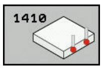
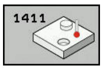
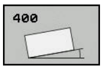
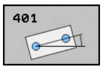
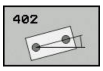
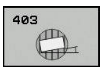
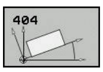
Abbr.	Inputs	Dialog
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool number	–
TYPE	Selection of the touch probe used	**Selection of the touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis in the principal axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis in the minor axis	**TS center misalignmt. aux. axis? [mm]**
CAL_ANG	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the control will probe the workpiece **F** can never exceed the value set in machine parameter **maxTouchFeed** (No. 122602).	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe is pre-positioning and is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within this defined value, the control will issue an error message.	**Maximum measuring range? [mm]**
SET_UP	In **set_up** you define how far from the defined or calculated touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to the **SET_UP** machine parameter.	**Set-up clearance? [mm]**
F_PREPOS	Defining speed with pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-position at rapid? ENT/ NOENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probing process. In this way, the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Probe oriented? Yes=ENT/ No=NOENT**
SERIAL	You need not make an entry in this column. The TNC automatically enters the serial number of the touch probe if the touch probe has an EnDat interface.	**Serial number?**
REACTION	Behavior in case of collision with the touch probe ■ **NCSTOP**: The NC program will be aborted. ■ **EMERGSTOP**: Emergency stop, safe braking of the axes.	**Reaction?**

15

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

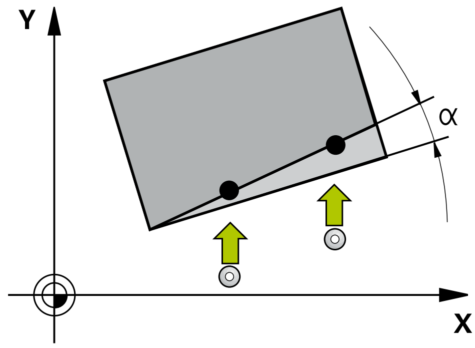
15.1 Overview

> The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.
>
> HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Cycle	Page
	1420 PROBING IN PLANE Automatic measurement using three points. Compensation via basic rotation.	550
	1410 PROBING ON EDGE Automatic measurement using two points. Compensation via basic rotation or a rotation of the rotary table.	554
	1411 PROBING TWO CIRCLES Automatic measurement using two cylindrical holes or studs. Compensation via basic rotation or a rotation of the rotary table.	558
	400 BASIC ROTATION Automatic measurement using two points. Compensation via basic rotation.	565
	401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.	568
	402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.	572
	403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation by turning the table.	577
	405 ROT IN C AXIS Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.	583
	404 SET BASIC ROTATION Setting any basic rotation.	582

15.2 Fundamentals of the 14xx touch probe cycles

Characteristics common to the 14xx touch probe cycles for measuring rotations



Rotations can be determined using the following three cycles:

- 1410 **PROBING ON EDGE**
- 1411 **PROBING TWO CIRCLES**
- 1420 **PROBING IN PLANE**

These cycles include the following functionality:

- Consideration of active machine kinematics
- Semi-automatic probing
- Monitoring of tolerances
- Consideration of 3-D calibration
- Rotation and position are measured simultaneously

The programmed positions are interpreted as nominal positions in the I-CS. The probing positions reference the programmed nominal coordinates.

Evaluation – preset:

- If you want to probe objects in a consistent machining plane or probe on position objects while the TCPM is active, you can program any required shifts as basic transformations in the preset table.
- In the basic transformations, rotations can be programmed as basic rotations or as axial offsets from the first rotary table axis, seen from the workpiece.

Logging:

The determined results are recorded in the **TCHPRAUTO.html** file and stored in the Q-parameter programmed for this cycle.
The measured deviations refer to the mean value of the tolerance range. If you do not specify tolerances, they refer to the nominal dimension.

> If you want to use not only the measured rotation, but also a measured position, make sure to probe the surface in its surface normal. The larger the angular error and the bigger the ball-tip radius, the larger the positioning error. If the angular errors in the initial angular position are too large, corresponding position errors might be the result.
>
> When probing with TCPM, existing 3-D calibration data are taken into account. If these calibration data do not exist, deviations might be the result.

Semi-automatic mode

If the workpiece placement has not been defined yet, you can use semi-automatic mode. In this mode, it is possible to determine the starting position by manually pre-positioning before you perform the probing operation on the desired object. This interruption is only possible in the machine operating modes, i.e. not in the Test run operating mode.

To do so, press the **ENTER TEXT** soft key and precede the nominal value of every coordinate of the object with a "**?**". If no nominal position has been defined, the control will perform an actual-to-nominal value transfer after probing the object. This means that the measured actual position will subsequently be applied as the nominal position. Consequentially, there is no deviation for this position and thus no position compensation. This can be advantageously used to avoid corrections of the preset for directions that have not been defined accurately in a semi-automatic program run.

Cycle run:

- The cycle interrupts the normal program run.
- A dialog window opens.
- Use the axis-direction keys or the handwheel to pre-position the touch probe at the desired point.
- If required, change the probing conditions, such as the probing direction.
- Press **NC start**.**NC start**
- Make sure that the touch probe returns to a safe position at the end of the cycle to avoid any collision in the further program run.

NOTICE

Danger of collision!

Depending on the object to be probed, the control will ignore the programmed mode for retraction to clearance height when running in semi-automatic mode. If semi-automatic mode has only been programmed for a single object to be probed, the cycle will ignore the programmed retraction to clearance height for this object only.

- Make sure that the touch probe will return to a safe position at the end of the cycle.

Example:

When aligning the touch probe to an edge with 0° in Cycle 1410, presetting is to be performed in the reference axis direction, but not in the secondary and tool axes as these probing positions have not been defined accurately.

5 TCH PROBE 1410 PROBING TWO CIRCLES		Define the cycle
QS1100= "?10"	**;1ST POINT REF AXIS**	Nominal position 1 of reference axis exists, position of workpiece unknown
QS1101= "?"	**;1ST POINT MINOR AXIS**	Nominal position 1 of minor axis unknown
QS1102= "?"	**;1ST POINT TOOL AXIS**	Nominal position 1 of tool axis unknown
QS1103= "?50"	**;2ND POINT REF AXIS**	Nominal position 2 of reference axis exists, position of workpiece unknown
QS1104= "?"	**;2ND POINT MINOR AXIS**	Nominal position 2 of minor axis unknown
QS1105= "?"	**;2ND POINT TOOL AXIS**	Nominal position 2 of tool axis unknown
Q372=+1	**;PROBING DIRECTION**	Probing direction (-3 to +3)
...	**;**	

Evaluation of tolerances

Optionally, the control can monitor the touch probe system for tolerances, distinguishing between the position and dimension of an object.

If you added a tolerance to a dimension, the control will monitor it and set an error status in the **Q183** return parameter if the tolerance is not met. Tolerance monitoring and the status always refer to the situation during probing, i.e. before the preset is corrected by the cycle.

Cycle run:

- If reaction to errors has been enabled (**Q309=1**), the control will check for scrap and rework. If scrap is found, the execution of the NC program will be interrupted. If **Q309=2**, then the control will only check for scrap. If scrap is found, the execution of the NC program will be interrupted
- If your workpiece is considered as scrap, a dialog window opens. It shows all nominal and measured values of the object.
- You can then decide whether to continue machining or abort the program. To resume the program run, press **NC start**; to abort the program, press the **CANCEL** soft key.**CANCEL**

> Please keep in mind that the touch probe cycles return the deviations with respect to the mean tolerance value in Q parameters **Q98x** and **Q99x**. Thus, these values equal the compensation values used by the cycle if input parameters **Q1120** and **Q1121** have been set accordingly. If no automatic evaluation has been programmed, it is thus easier to use these values, that reference the mean tolerance value, for other purposes.

5 TCH PROBE 1410 PROBING TWO CIRCLES		Define the cycle
Q1100=+50	**;1ST POINT REF AXIS**	Nominal position 1 of reference axis
Q1101= +50	**;1ST POINT MINOR AXIS**	Nominal position 1 of minor axis
Q1102= -5	**;1ST POINT TOOL AXIS**	Nominal position 1 of tool axis
QS1116="+9-1-0.5"	**;DIAMETER 1**	Diameter 1, tolerance specified
Q1103= +80	**;2ND POINT REF AXIS**	Nominal position 2 of reference axis
Q1104=+60	**;2ND POINT MINOR AXIS**	Nominal position 2 of minor axis
QS1105= -5	**;2ND POINT TOOL AXIS**	Nominal position 2 of tool axis
QS1117="+9-1-0,5"	**;DIAMETER 2**	Diameter 2, tolerance specified
...	**;**	
Q309=2	**;ERROR REACTION**	
...	**;**	

Transferring the actual position

You can determine the actual position in advance and define it as the actual position for the touch probe cycle. Then, both the nominal position and the actual position will be transferred. Based on the difference, the cycle calculates the required compensation values and applies tolerance monitoring.

Please note that in this case, the control does not perform probing, but only accounts for the actual and nominal positions.

To do so, press the **ENTER TEXT** soft key and follow the nominal value of each coordinate in the object with a "**@**". You can then enter the actual position after the "**@**".

> You must define the actual position for all three axes: reference axis, minor axis, and tool axis. If you define only one axis with its actual position, an error message will be generated.
>
> Actual positions can also be defined with Q parameters **Q1900-Q1999**.

Example:

This feature allows you to do the following:

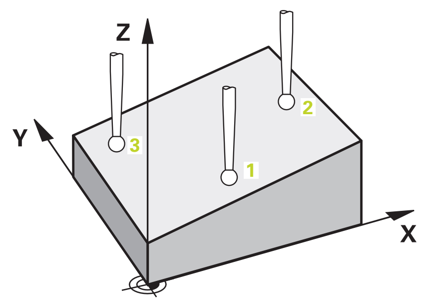
- Determine a circular pattern based on multiple different objects
- Align a gear wheel via its center and the position of a tooth

5 TCH PROBE 1410 PROBING ON EDGE	
QS1100= "10+0.02@10.0123"	
;1ST POINT REF AXIS	Nominal position 1 of the reference axis with tolerance monitoring and actual position
QS1101="50@50.0321"	
;1ST POINT MINOR AXIS	Nominal position 1 of the minor axis and actual position
QS1102= "-10-0.2+0.02@Q1900"	
;1ST POINT TOOL AXIS	Nominal position 1 of the tool axis with tolerance monitoring and actual position
... ;	

15.3 PROBING IN PLANE (Cycle 1420, ISO: G1420)

Cycle run

Touch probe cycle 1420 finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.



1 The control positions the touch probe at rapid traverse (value from the **FMAX** column), using positioning logic "Executing touch probe cycles", to the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
2 The touch probe returns to the clearance height (depends on **Q1125**), then moves in the working plane to touch point **2** and measures the actual value of the second point of the plane.
3 The touch probe returns to the clearance height (depends on **Q1125**), then moves in the working plane to touch point **3** and measures the actual value of the third point of the plane.
4 Finally the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:

Parameter number	Meaning
Q950 to Q952	Measured position 1 in the principal, secondary, and tool axes
Q953 to Q955	Measured position 2 in the principal, secondary, and tool axes
Q956 to Q958	Measured position 3 in the principal, secondary, and tool axes
Q961 to Q963	Measured spatial angle SPA, SPB, and SPC in the WP-CS
Q980 to Q982	Measurement error 1 of the positions: principal, secondary, and tool axis
Q983 to Q985	Measurement error 2 of the positions: principal, secondary, and tool axis
Q986 to Q988	Measurement error 3 of the positions: principal, secondary, and tool axis
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

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

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

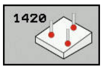
Alignment with rotary axes is only possible if two rotary axes have been defined in the kinematics.

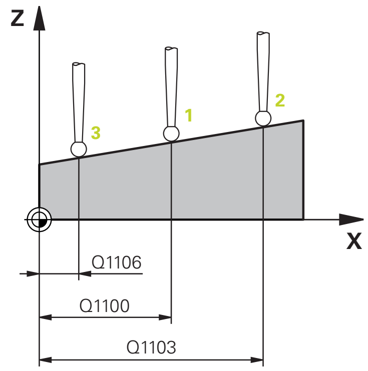
If **Q1121** equals 0 and **Q1126** is not equal to 0, an error message will be displayed because this would align the rotary axes, but the rotation would not be evaluated.

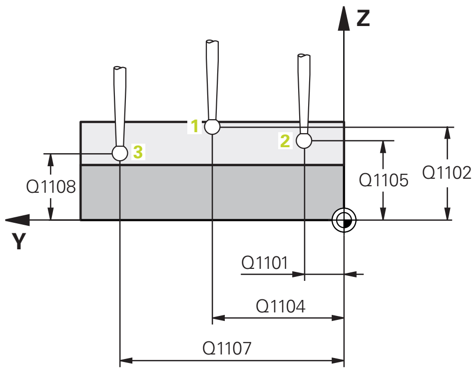
The deviations are the differences of the measured actual values to the mean tolerance value, not the difference to the nominal value.

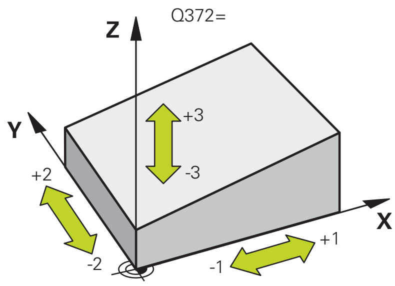
The saved spatial angle is saved in parameters **Q961** to **Q963**. Based on the definition of the nominal positions, you can define the nominal spatial angle. The difference between the measured spatial angle and the nominal spatial angle is used to transfer the basic rotation in 3-D in the preset table.

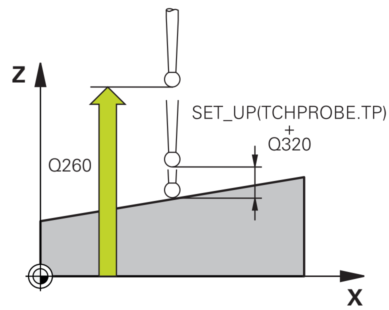
Cycle parameters











- **Q1100 1st noml. position of ref. axis** (absolute): Nominal coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal coordinate of the first touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1103 2nd noml. position of ref axis?** (absolute): Nominal coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): Nominal coordinate of the second touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1106 3rd noml. position of ref axis?** (absolute): Nominal coordinate of the third touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1107 3rd noml. position minor axis?** (absolute): Nominal coordinate of the third touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1108 3rd nominal position tool axis?** (absolute): Nominal coordinate of the third touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input range: -3 to +3
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999

- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the measuring points:
  -1: Do not move to clearance height
  0: Move to clearance height before and after the cycle
  1: Move to clearance height before and after moving to each measured object
  2: Move to clearance height before and after moving to each measuring point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt program run, do not display an error message
  1: If the tolerance is exceeded, interrupt program run and display an error message
  2: If the determined actual coordinate indicates that the workpiece is scrap, the control displays a message and interrupts program run. In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.
- **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
  0: Maintain the current tilting axes position
  1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the ball tip (TURN)
- **Q1120 Transfer position?**: Define which measured actual position is to be transfered to the preset table by the control:
  0: Do not transfer any positions
  1: Transfer measuring point 1
  2: Transfer measuring point 2
  3: Transfer measuring point 3
  4: Transfer an averaged measuring point
- **Q1121 Confirm basic rotation?**: Define whether the control is to transfer the determined inclination as the basic rotation:
  0: No basic rotation
  1: Set basic rotation: The control saves the basic rotation.

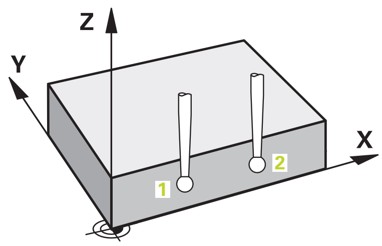
Example

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

15.4 PROBING ON EDGE (Cycle 1410, ISO: G1410)

Cycle run

Touch probe cycle 1410 measures the angle that any straight line on the workpiece forms with the reference axis of the working plane.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", to the programmed touch point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe in the direction opposite to the direction of probing.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:

Parameter number	Meaning
Q950 to Q952	Measured position 1 in the principal, secondary, and tool axes
Q953 to Q955	Measured position 2 in the principal, secondary, and tool axes
Q964	Measured angle of rotation in the IP-CS
Q965	Measured angle of rotation in the coordinate system of the rotary table
Q980 to Q982	Measurement error 1 of the positions: principal, secondary, and tool axis
Q983 to Q985	Measurement error 2 of the positions: principal, secondary, and tool axis
Q994	Measured angle deviation in the IP-CS
Q995	Measured angle deviation in the coordinate system of the rotary table
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

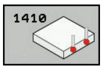
Before defining this cycle, you must have programmed a tool call to define the touch probe axis. This touch probe axis must be the Z axis.

Alignment with rotary axes is only possible if the measured rotation can be compensated using a rotary table axis, which is the first rotary table axis, seen from the workpiece.

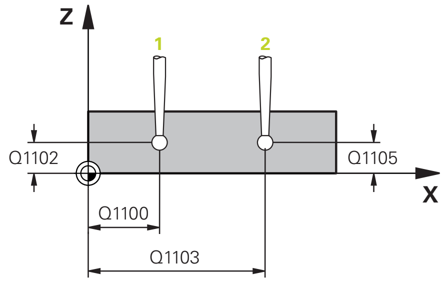
If **Q1121** is not equal to 2 and **Q1126** is not equal to 0, an error message will be displayed. It would be contradictory to align the rotary axis and at the same time activate a basic rotation.

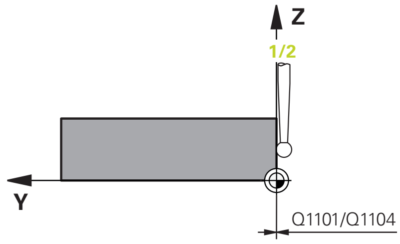
The deviations are the differences of the measured actual values to the mean tolerance value (including the tolerance factor), not the difference to the nominal value.

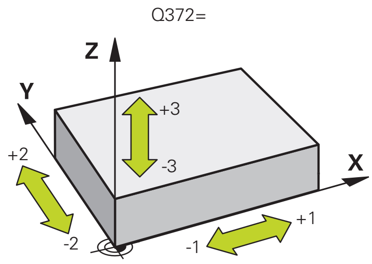
Cycle parameters

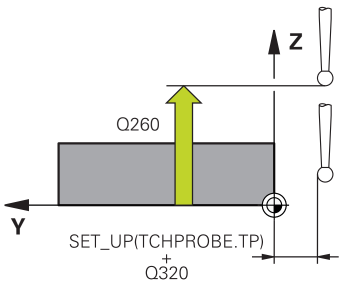


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal coordinate of the first touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1103 2nd noml. position of ref axis?** (absolute): Nominal coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): Nominal coordinate of the second touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input range: -3 to +3
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999









- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the measuring points:
  -1: Do not move to clearance height
  0: Move to clearance height before and after the cycle
  1: Move to clearance height before and after moving to each measured object
  2: Move to clearance height before and after moving to each measuring point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt program run, do not display an error message
  1: If the tolerance is exceeded, interrupt program run and display an error message
  2: If the determined actual coordinate indicates that the workpiece is scrap, the control displays a message and interrupts program run. In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.
- **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
  0: Maintain the current tilting axes position
  1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the ball tip (TURN)
- **Q1120 Transfer position?**: Define which measured actual position is to be transfered to the preset table by the control:
  0: Do not transfer any positions
  1: Transfer measuring point 1
  2: Transfer measuring point 2
  3: Transfer an averaged measuring point
- **Q1121 CONFIRM ROTATION?**: Define whether the control is to transfer the determined inclination as the basic rotation:
  0: No basic rotation
  1: Set basic rotation: The control saves the basic rotation.
  2: Perform rotation of the rotary table: The control will make an entry in the corresponding **Offset** column of the preset table

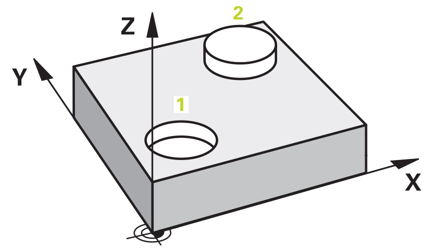
Example

5 TCH PROBE 1410 PROBING ON EDGE	
Q1100=+0	;1ST POINT REF AXIS
Q1101=+0	;1ST POINT MINOR AXIS
Q1102=+0	;1ST POINT TOOL AXIS
Q1103=+0	;2ND POINT REF AXIS
Q1104=+0	;2ND POINT MINOR AXIS
Q1105=+0	;2ND POINT TOOL AXIS
Q372=+1	;PROBING DIRECTION
Q320=+0	;SET-UP CLEARANCE
Q260=+100	;CLEARANCE HEIGHT
Q1125=+2	;CLEAR. HEIGHT MODE
Q309=+0	;ERROR REACTION
Q1126=+0	;ALIGN ROTARY AXIS
Q1120=+0	;TRANSER POSITION
Q1121=+0	;CONFIRM ROTATION

15.5 PROBING TWO CIRCLES (Cycle 1411, ISO G1411)

Cycle run

Touch probe cycle 1411 measures the centers of two holes or cylindrical studs. Then the control calculates the angle between the reference axis in the working plane and the line connecting the hole or stud center points. With the basic rotation function, the control compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.



1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic "Executing touch probe cycles", to the programmed center point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
2 Then the probe moves to the entered measuring height and probes (depending on the number of probes in **Q423**) the first hole or stud center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole or second stud **2**.
4 Then the control moves the probe to the entered measuring height and probes (depending on the number of probes in **Q423**) the second hole or stud center point.
5 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:

Parameter number	Meaning
Q950 to Q952	Measured position 1 in the principal, secondary, and tool axes
Q953 to Q955	Measured position 2 in the principal, secondary, and tool axes
Q964	Measured angle of rotation in the IP-CS
Q965	Measured angle of rotation in the coordinate system of the rotary table
Q966 to Q967	Measured first and second diameters
Q980 to Q982	Measurement error 1 of the positions: principal, secondary, and tool axis
Q983 to Q985	Measurement error 2 of the positions: principal, secondary, and tool axis
Q994	Measured angle deviation in the IP-CS

Parameter number	Meaning
Q995	Measured angle deviation in the coordinate system of the rotary table
Q996 to Q997	Measurement error of the first and second diameters
Q183	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

Please note while programming!

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

Alignment with rotary axes is only possible if the measured rotation can be compensated using a rotary table axis, which is the first rotary table axis, seen from the workpiece.

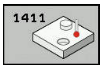
If **Q1121** is not equal to 2 and **Q1126** is not equal to 0, an error message will be displayed. It would be contradictory to align the rotary axis and at the same time activate a basic rotation.

The deviations are the differences of the measured actual values to the mean tolerance value, not the difference to the nominal value.

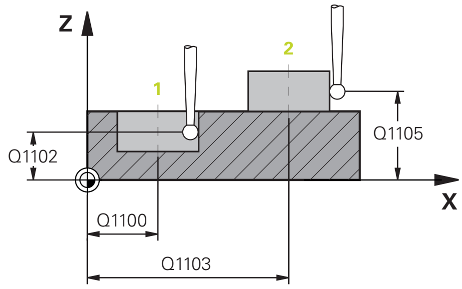
If the hole diameter is less than the ball tip diameter, an error message will be displayed.

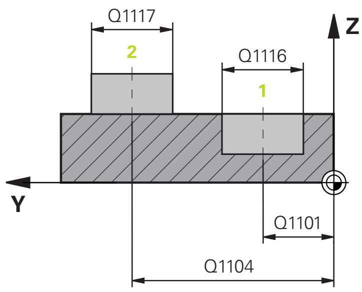
Is the hole diameter is so small that the programmed set-up clearance cannot be achieved, a dialog opens. The dialog shows the nominal value corresponding to the hole radius, the calibrated ball-tip radius and the achievable set-up clearance. Either confirm the dialog with **NC start** or cancel the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

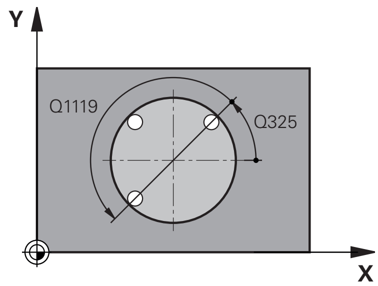
Cycle parameters

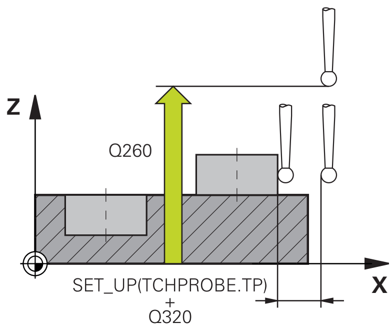


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1102 1st nominal position tool axis?** (absolute): Nominal coordinate of the first touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1116 Diameter of 1st position?**: Diameter of the first hole or stud. Input range: 0 to 9999.9999
- **Q1103 2nd noml. position of ref axis?** (absolute): Nominal coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1104 2nd noml. position of minor axis** (absolute): Nominal coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1105 2nd nominal pos. of tool axis?** (absolute): Nominal coordinate of the second touch point in the tool axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q1117 Diameter of 2nd position?**: Diameter of the second hole or stud. Input range: 0 to 9999.9999
- **Q1115 Geometry type (0-3)?**: Define the geometry of the objects to be probed
  0: Position 1 = hole and position 2 = hole
  1: Position 1 = stud and position 2 = stud
  2: Position 1 = hole and position 2 = stud
  3: Position 1 = stud and position 2 = hole
- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000









- **Q1119 Arc angular length?**: Angular range in which the touch points are distributed. Input range: -359.999 to +360
- **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q1125 Traverse to clearance height?**: Define the behavior of the touch probe between the measuring points:
  -1: Do not move to clearance height
  0: Move to clearance height before and after the cycle
  1: Move to clearance height before and after moving to each measured object
  2: Move to clearance height before and after moving to each measuring point
- **Q309 Reaction to tolerance error?**: Specify whether the control is to interrupt program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not interrupt program run, do not display an error message
  1: If the tolerance is exceeded, interrupt program run and display an error message
  2: If the determined actual coordinate indicates that the workpiece is scrap, the control displays a message and interrupts program run. In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.
- **Q1126 Align rotary axes?**: Position the tilting axes for inclined machining:
  0: Maintain the current tilting axes position
  1: Position the tilting axis automatically and orient the ball tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  2: Position the tilting axis automatically without orienting the ball tip (TURN)
- **Q1120 Transfer position?**: Define which measured actual position is to be transfered to the preset table by the control:
  0: Do not transfer any positions
  1: Transfer measuring point 1
  2: Transfer measuring point 2
  3: Transfer an averaged measuring point

Example

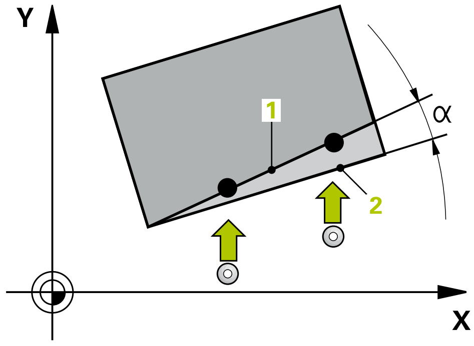
5 TCH PROBE 1410 PROBING TWO CIRCLES	
Q1100=+0	;1ST POINT REF AXIS
Q1101=+0	;1ST POINT MINOR AXIS
Q1102=+0	;1ST POINT TOOL AXIS
Q1116=0	;DIAMETER 1
Q1103=+0	;2ND POINT REF AXIS
Q1104=+0	;2ND POINT MINOR AXIS
Q1105=+0	;2ND POINT TOOL AXIS
Q1117=+0	;DIAMETER 2
Q1115=0	;GEOMETRY TYPE
Q423=4	;NO. OF PROBE POINTS
Q325=+0	;STARTING ANGLE
Q1119=+360	;ANGULAR LENGTH
Q320=+0	;SET-UP CLEARANCE
Q260=+100	;CLEARANCE HEIGHT
Q1125=+2	;CLEAR. HEIGHT MODE
Q309=+0	;ERROR REACTION
Q1126=+0	;ALIGN ROTARY AXIS
Q1120=+0	;TRANSER POSITION
Q1121=+0	;CONFIRM ROTATION

- **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 Fundamentals of the 4xx touch probe cycles

Characteristics common to all touch probe cycles for measuring workpiece misalignment

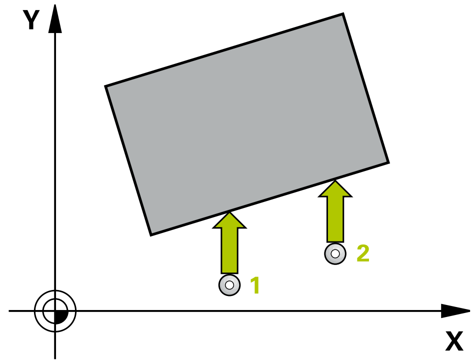
For Cycles 400, 401 and 402 you can define through parameter **Q307 Preset value for rotation angle** whether the measurement result is to be corrected by a known angle # (see figure at right). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.



> These cycles do not work with 3-D Rot! In such a case, use Cycles 14xx. **Further information:** "Fundamentals of the 14xx touch probe cycles ", Page 545

15.7 BASIC ROTATION (Cycle 400, ISO: G400)

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 539), 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:

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

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

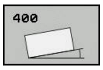
NOTICE

Danger of collision!

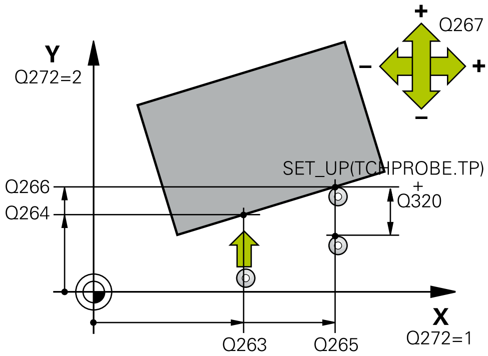
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

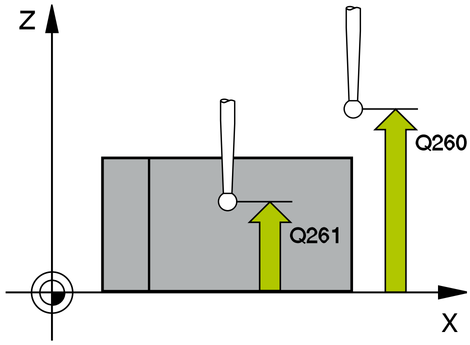
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





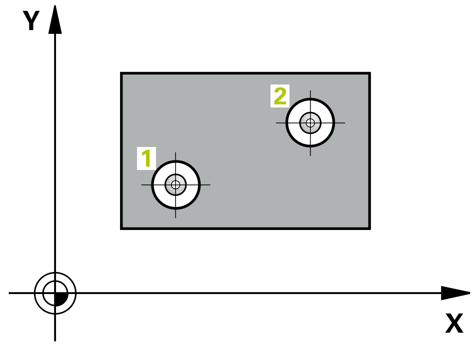
Example

5 TCH PROBE 400 BASIC ROTATION	
Q263=+10	;1ST POINT 1ST AXIS
Q264=+3,5	;1ST POINT 2ND AXIS
Q265=+25	;2ND POINT 1ST AXIS
Q266=+2	;2ND PNT IN 2ND AXIS
Q272=+2	;MEASURING AXIS
Q267=+1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q307=0	;PRESET ROTATION ANG.
Q305=0	;NUMBER IN TABLE

- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the reference axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: -360.000 to 360.000
- **Q305 Preset number in table?**: Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. If you enter Q305=0, the control automatically places the determined basic rotation in the ROT menu of the Manual Operation mode. Input range: 0 to 99999

15.8 BASIC ROTATION over two holes (Cycle 401, ISO: G401)

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 539), 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:

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

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

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

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

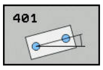
NOTICE

Danger of collision!

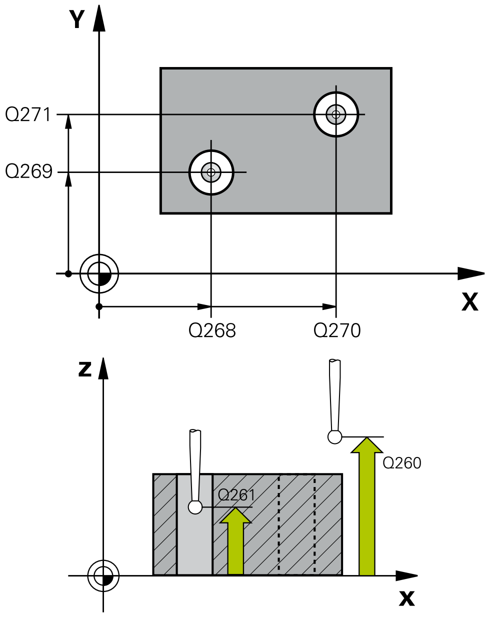
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

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

Cycle parameters



- **Q268 1st hole: center in 1st axis?** (absolute): Center of the first hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q269 1st hole: center in 2nd axis?** (absolute): Center of the first hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q270 2nd hole: center in 1st axis?** (absolute): Center of the second hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q271 2nd hole: center in 2nd axis?** (absolute): Center of the second hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999



Example

```
5 TCH PROBE 401  ROT OF 2 HOLES
    Q268=-37    ;1ST CENTER 1ST AXIS
    Q269=+12    ;1ST CENTER 2ND AXIS
    Q270=+75    ;2ND CENTER 1ST AXIS
```

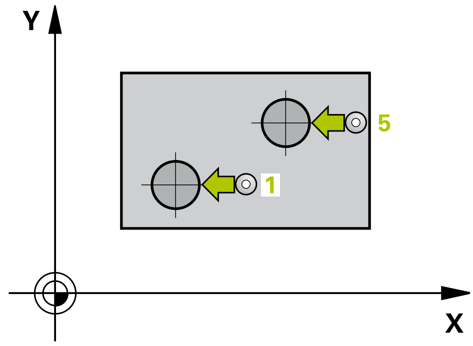
- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the reference axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: -360.000 to 360.000
- **Q305 Number in table?** Enter the number of a row in the preset table. The control will enter the value in this row: Input range: 0 to 99999
  Q305 = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.
  Q305 > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**).
  Q305 depends on the following parameters:
  Q337 = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in Q305. (Example: For tool axis Z, an entry is made in column **SPC**)
  Q337 = 0 and, at the same time, **Q402** = 1: Parameter Q305 is not effective
  Q337 = 1 Parameter Q305 is in effect as described above
- **Q402 Basic rotation/alignment (0/1)**: Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of the rotary table:
  0: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**)
  1: Rotate the rotary table: An entry will be made in the **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column), in addition, the corresponding axis will be rotated
- **Q337 Set to zero after alignment?**: Define whether the control is to set the position display of the corresponding rotary axis to 0 after the alignment:
  0: The position display is not set to 0 after the alignment
  1: After the alignment, the position display is set to 0, provided you defined **Q402=1**

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

15.9 BASIC ROTATION over two studs (Cycle 402, ISO: G402)

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 539), 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:

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

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

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

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

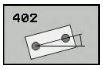
NOTICE

Danger of collision!

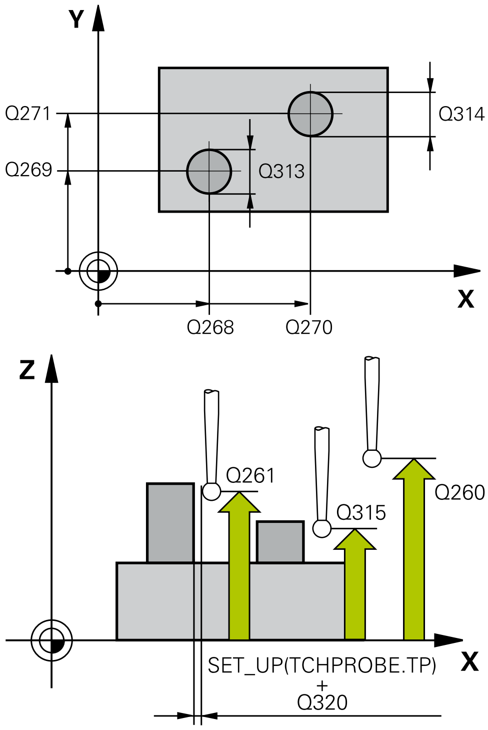
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

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

Cycle parameters



- **Q268 1st stud: center in 1st axis?** (absolute): Center of the first stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q269 1st stud: center in 2nd axis?** (absolute): Center of the first stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q313 Diameter of stud 1?**: Approximate diameter of the first stud. Enter a value that is more likely to be too large than too small. Input range: 0 to 99999.9999
- **Q261 Meas. height stud 1 in TS axis?** (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 1 is to be measured. Input range: -99999.9999 to 99999.9999
- **Q270 2nd stud: center in 1st axis?** (absolute): Center of the second stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q271 2nd stud: center in 2nd axis?** (absolute): Center of the second stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q313 Diameter of stud 2?**: Approximate diameter of the second stud. Enter a value that is more likely to be too large than too small. Input range: 0 to 99999.9999
- **Q315 Meas. height stud 2 in TS axis?** (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 2 is to be measured. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points



Example

5 TCH PROBE 402 ROT OF 2 STUDS	
Q268=-37	;1ST CENTER 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q313=60	;DIAMETER OF STUD 1
Q261=-5	;MEAS. HEIGHT STUD 1
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q314=60	;DIAMETER OF STUD 2
Q315=-5	;MEAS. HEIGHT STUD 2
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q307=0	;PRESET ROTATION ANG.
Q305=0	;NUMBER IN TABLE
Q402=0	;COMPENSATION
Q337=0	;SET TO ZERO

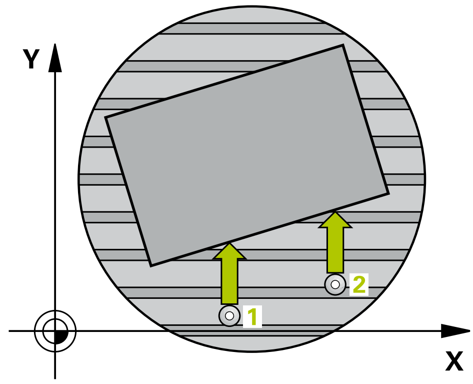
- **Q307 Preset value for rotation angle** (absolute): If the misalignment is to be measured against any straight line rather than the reference axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation. Input range: -360.000 to 360.000
- **Q305 Number in table?** Enter the number of a row in the preset table. The control will enter the value in this row: Input range: 0 to 99999
  Q305 = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.
  Q305 > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**).
  Q305 depends on the following parameters:
  Q337 = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in Q305. (Example: For tool axis Z, an entry is made in column **SPC**)
  Q337 = 0 and, at the same time, **Q402** = 1: Parameter Q305 is not effective
  Q337 = 1 Parameter Q305 is in effect as described above

- **Q402 Basic rotation/alignment (0/1)**: Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of the rotary table:
  0: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**)
  1: Rotate the rotary table: An entry will be made in the **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column), in addition, the corresponding axis will be rotated
- **Q337 Set to zero after alignment?**: Define whether the control is to set the position display of the corresponding rotary axis to 0 after the alignment:
  0: The position display is not set to 0 after the alignment
  1: After the alignment, the position display is set to 0, provided you defined **Q402=1**

15.10 BASIC ROTATION compensation via rotary axis (Cycle 403, ISO: G403)

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 539), to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and rotates the rotary axis, which was defined in the cycle, by the measured value. Optionally, you can specify whether the control is to set the determined rotation angle to 0 in the preset table or in the datum table.

Please note while programming:

NOTICE

Danger of collision!

If the control positions the rotary axis automatically, a collision might occur.

- Check for possible collisions between the tool and any elements positioned on the table
- Select the clearance height to prevent collisions

NOTICE

Danger of collision!

If you set parameter Q312 Axis for compensating movement? to 0, then the cycle will automatically determine the rotary axis to be aligned (recommended setting). When doing so, it determines an angle that depends on the sequence of the touch points. The measured angle goes from the first to the second touch point. If you select the A, B, or C axis as compensation axis in parameter Q312, the cycle determines the angle, regardless of the sequence of the touch points. The calculated angle lies in the range from -90° to +90°.

- After alignment, check the position of the rotary axis.

NOTICE

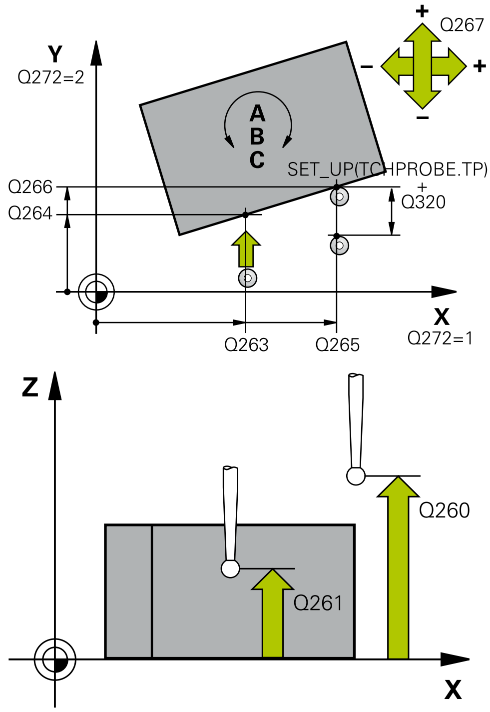
Danger of collision!

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

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

Cycle parameters

- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points



Example

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

- **Q312 Axis for compensating movement?**: Specify the rotary axis in which the control is to compensate the measured misalignment:
  0: Automatic mode—the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting!
  4: Compensate misalignment with rotary axis A
  5: Compensate misalignment with rotary axis B
  6: Compensate misalignment with rotary axis C
- **Q337 Set to zero after alignment?**: Define whether the control should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment.
  0: Do not set the angle of the rotary axis to 0 in the table after alignment
  1: Set the angle of the rotary axis to 0 in the table after alignment
- **Q305 Number in table?** Specify the number of the row in the preset table in which the control is to enter the basic rotation. Input range: 0 to 99999
  Q305 = 0: The rotary axis is zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.
  Q305 > 0: Specify the number of the row in the preset table in which the control is to zero the rotary axis. The control will make an entry in the **OFFSET** column of the preset table.
  Q305 depends on the following parameters:
  Q337 = 0: Parameter Q305 is not effective
  Q337 = 1: Parameter Q305 is effective as described above
  Q312 = 0: Parameter Q305 is effective as described above
  Q312 > 0: The entry in Q305 is ignored. The control will make an entry in the **OFFSET** column, in the row of the preset table that was active when the cycle was called.

- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
  0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q380 Ref. angle in ref. axis?**: Angle to which the control is to align the probed straight line. Only effective if the rotary axis is in automatic mode or if C is selected (Q312 = 0 or 6). Input range: -360.000 to 360.000

15.11 SET BASIC ROTATION (Cycle 404, ISO: G404)

Cycle run

With touch probe cycle 404, you can set any basic rotation automatically during program run or save it to the preset table. You can also use Cycle 404 if you want to reset an active basic rotation.

NOTICE

Danger of collision!

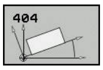
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

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

Example

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

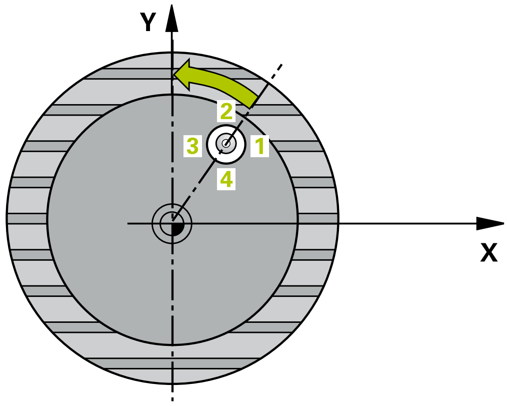
Cycle parameters

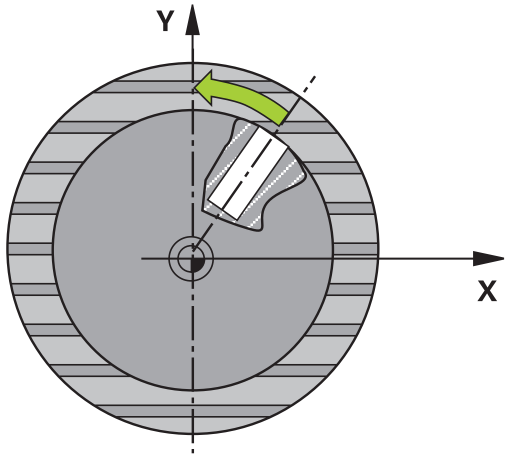


- **Q307 Preset value for rotation angle**: Angular value to which the basic rotation is to be set. Input range: -360.000 to 360.000
- **Q305 Preset number in table?**: Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. Input range: -1 to 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 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.

15.12 Compensating workpiece misalignment by rotating the C axis (Cycle 405, ISO: G405)

Cycle run





With touch probe cycle 405, you can measure

- the angular offset between the positive Y axis of the active coordinate system and the center line of a hole, or
- the angular offset between the nominal position and the actual position of a hole center.

The control compensates the determined angular offset by rotating the C axis. The workpiece can be clamped in any position on the rotary table, but the Y coordinate of the hole must be positive. If you measure the angular misalignment of the hole with touch probe axis Y (horizontal position of the hole), it may be necessary to execute the cycle more than once because the measuring strategy causes an inaccuracy of approx. 1% of the misalignment.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the hole center measured.
5 Finally the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center after compensation lies in the direction of the positive Y axis, or at the nominal position of the hole center—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in parameter Q150.

Please note while programming:

- Before a cycle definition you must have programmed a tool call to define the touch probe axis.
- The smaller the stepping angle, the less accurately the control can calculate the circle center point. Minimum input value: 5°

NOTICE

Danger of collision!

If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

- Make sure that there is no material in the pocket/hole
- To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

NOTICE

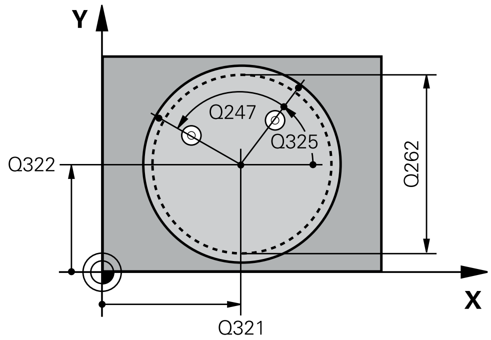
Danger of collision!

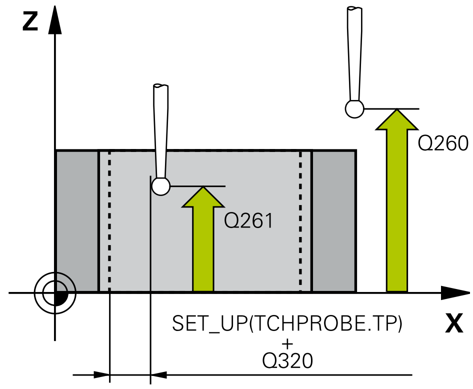
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

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

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the hole in the minor axis of the working plane. If you program Q322 = 0, the control aligns the hole center point with the positive Y axis. If you program Q322 # 0, then the control aligns the hole center point with the nominal position (angle resulting from the position of the hole center). Input range: -99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999



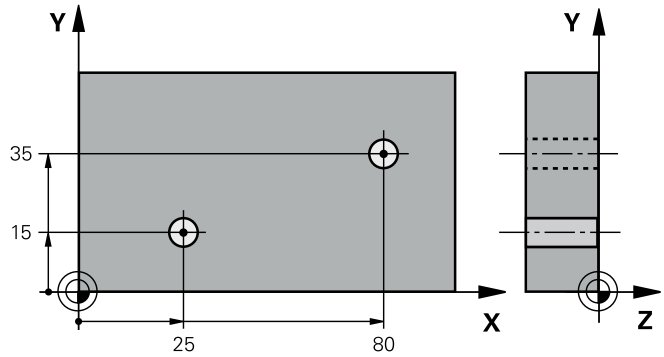


Example

5 TCH PROBE 405 ROT IN C AXIS	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=10	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=90	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q337=0	;SET TO ZERO

- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q337 Set to zero after alignment?**:
  0: Set the display of the C axis to 0 and write value to **C_Offset** of the active row of the datum table
  >0: Write the measured angular offset to the datum table. Row number = value in Q337. If a C-axis shift is registered in the datum table, the control adds the measured angular offset with the correct sign, positive or negative.

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

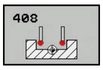
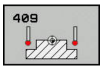
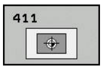
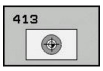
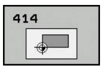
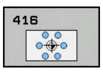
Touch Probe Cycles: Automatic Presetting

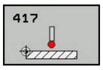
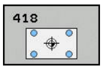
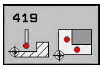
16.1 Fundamentals

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
	408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as preset	593
	409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as preset	597
	410 DATUM INSIDE RECTANGLE Measuring the inside length and width of a rectangle, and defining the center as preset	601
	411 DATUM OUTSIDE RECTANGLE Measuring the outside length and width of a rectangle, and defining the center as preset	605
	412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as preset	609
	413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as preset	614
	414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as preset	619
	415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as preset	624
	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	629

Soft key	Cycle	Page
	417 PRESET IN TS AXIS (2nd soft-key row) Measuring any position in the touch probe axis and defining it as preset	634
	418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as preset	636
	419 DATUM IN ONE AXIS (2nd soft-key row) Measuring any position in any axis and defining it as preset	641

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

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

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

NOTICE

Danger of collision!

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

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

Characteristics common to all touch probe cycles for presetting

> You can also run the Touch Probe Cycles 408 to 419 during an active rotation (basic rotation or Cycle 10).

Preset and touch probe axis

The control determines the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Set preset in
Z	X and Y
Y	Z and X
X	Y and Z

Saving the calculated preset

In all cycles for presetting, you can use the input parameters Q303 and Q305 to define how the control is to save the calculated preset:

- **Q305 = 0, Q303 = 1:**
  The control copies the active preset to row 0 and activates row 0. This will delete simple transformations.
- **Q305 not equal to 0, Q303 = 0:**
  The result is written to the datum table, row **Q305**. **Activate the datum with Cycle 7 in the NC program.**
- **Q305 not equal to 0, Q303 = 1:**
  The control writes the result to the preset table, row **Q305**. The reference system is the machine coordinate system (REF coordinates). **Activate the preset with Cycle 247 in the NC program.**
- **Q305 not equal to 0, Q303 = -1**

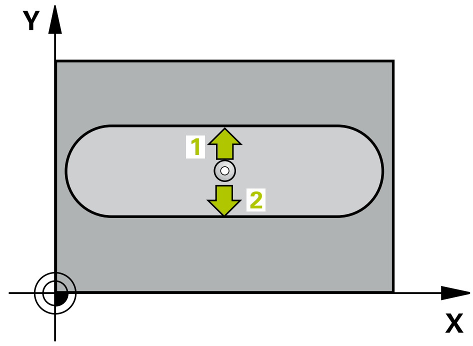
> This combination can only occur if you
> - read in NC programs containing Cycles 410 to 418 created on a TNC 4xx
> - read in NC programs containing Cycles 410 to 418 created with an older software version on an iTNC 530
> - did not specifically define the measured-value transfer with parameter Q303 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 SLOT CENTER PRESET (Cycle 408, ISO: G408)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the actual values in the Q parameters listed below.
5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Please note while programming:

NOTICE

Danger of collision!

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

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

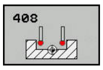
NOTICE

Danger of collision!

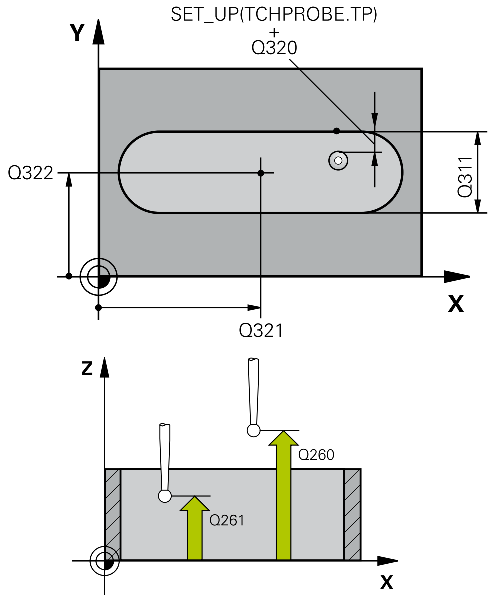
To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width. If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch probe does not return to the clearance height between the two measuring points.

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

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the slot in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the slot in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q311 Width of slot?** (incremental): Width of the slot, regardless of its position in the working plane. Input range: 0 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated slot center. Default setting = 0. Input range: -99999.9999 to 99999.9999



Example

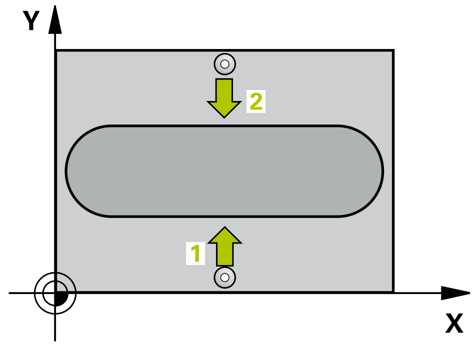
5 TCH PROBE 408 SLOT CENTER PRESET	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q311=25	;SLOT WIDTH
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=10	;NUMBER IN TABLE
Q405=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET

- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
  0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

16.3 RIDGE CENTER PRESET (Cycle 409, ISO: G409)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.
4 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the actual values in the Q parameters listed below.
5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

Please note while programming:

NOTICE

Danger of collision!

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

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

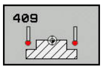
NOTICE

Danger of collision!

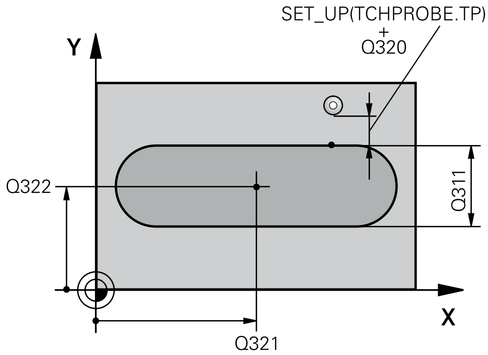
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

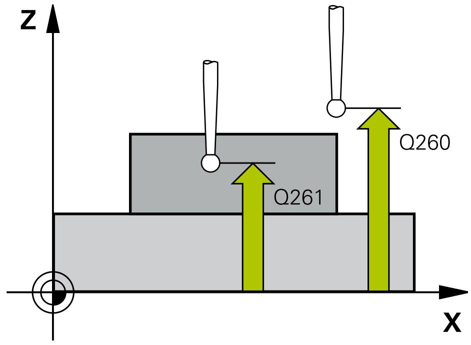
- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the ridge in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the ridge in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q311 Ridge width?** (incremental): Width of the ridge, regardless of its position in the working plane. Input range: 0 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.





Example

5 TCH PROBE 409 RIDGE CENTER PRESET	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q311=25	;RIDGE WIDTH
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q305=10	;NUMBER IN TABLE
Q405=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS

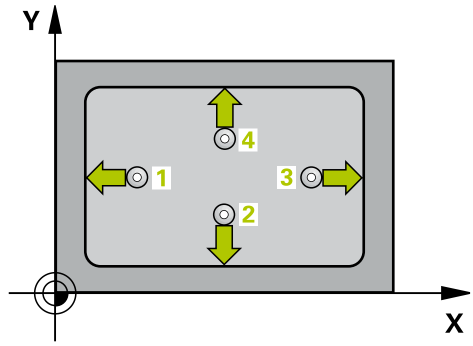
- **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated ridge center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:
  0: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET

16.4 PRESET FROM INSIDE OF RECTANGLE (Cycle 410, ISO: G410)

Cycle run

Touch probe cycle 410 finds the center of a rectangular pocket and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters Q303 and Q305. (see "Characteristics common to all touch probe cycles for presetting", Page 592)
6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Please note while programming:

NOTICE

Danger of collision!

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

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

NOTICE

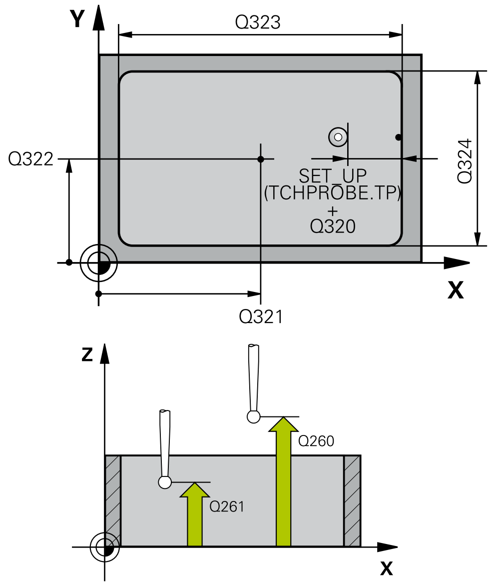
Danger of collision!

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

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

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the pocket in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q323 First side length?** (incremental): Pocket length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q324 Second side length?** (incremental): Pocket length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999



Example

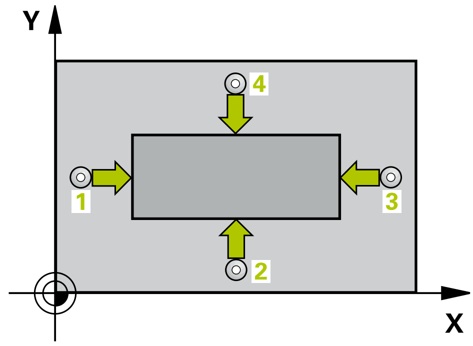
5 TCH PROBE 410 PRESET INSIDE RECTAN	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;FIRST SIDE LENGTH
Q324=20	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=10	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET

- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

16.5 PRESET FROM OUTSIDE OF RECTANGLE (Cycle 411, ISO: G411)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters Q303 and Q305. (see "Characteristics common to all touch probe cycles for presetting", Page 592)
6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Please note while programming:

NOTICE

Danger of collision!

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

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

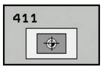
NOTICE

Danger of collision!

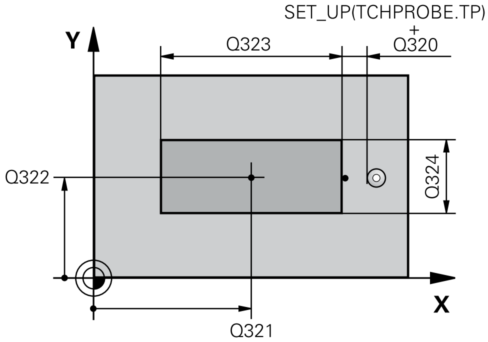
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

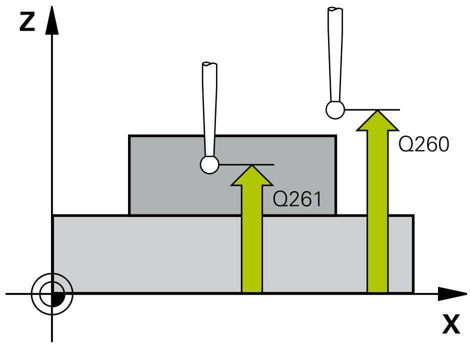
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q323 First side length?** (incremental): Stud length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q324 Second side length?** (incremental): Stud length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999





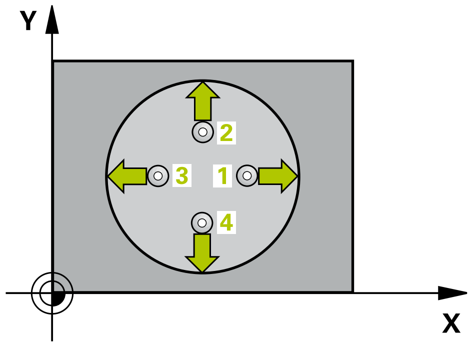
Example

5 TCH PROBE 411 PRESET OUTS. RECTAN	
Q321=+50	**;CENTER IN 1ST AXIS**
Q322=+50	**;CENTER IN 2ND AXIS**
Q323=60	**;FIRST SIDE LENGTH**
Q324=20	**;2ND SIDE LENGTH**
Q261=-5	**;MEASURING HEIGHT**
Q320=0	**;SET-UP CLEARANCE**
Q260=+20	**;CLEARANCE HEIGHT**
Q301=0	**;MOVE TO CLEARANCE**
Q305=0	**;NUMBER IN TABLE**
Q331=+0	**;PRESET**
Q332=+0	**;PRESET**
Q303=+1	**;MEAS. VALUE TRANSFER**
Q381=1	**;PROBE IN TS AXIS**
Q382=+85	**;1ST CO. FOR TS AXIS**
Q383=+50	**;2ND CO. FOR TS AXIS**
Q384=+0	**;3RD CO. FOR TS AXIS**
Q333=+1	**;PRESET**

- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

16.6 PRESET FROM INSIDE OF CIRCLE (Cycle 412, ISO: G412)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2. Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3. Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4. The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5. Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the actual values in the Q parameters listed below.
6. If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

- The smaller the stepping angle Q247, the less accurately the control can calculate the preset. Minimum input value: 5°
- Program a stepping angle to be less than 90°; input range -120° to 120°

NOTICE

Danger of collision!

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

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

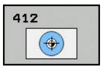
NOTICE

Danger of collision!

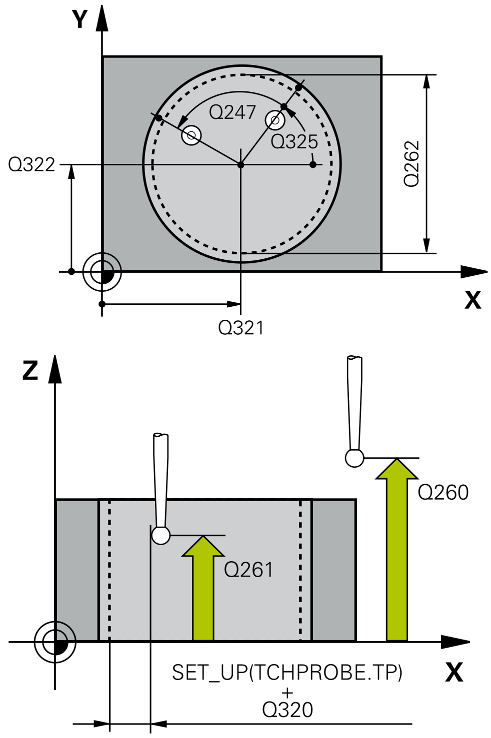
To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole). If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

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

Cycle parameters



- **Q321 Center in 1st axis?** (absolute): Center of the pocket in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the minor axis of the working plane. If you program Q322 = 0, the control aligns the hole center point to the positive Y axis. If you program Q322 not equal to 0, then the control aligns the hole center point to the nominal position. Input range: -99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999



Example

5 TCH PROBE 412 PRESET INSIDE CIRCLE	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE

- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the pocket center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

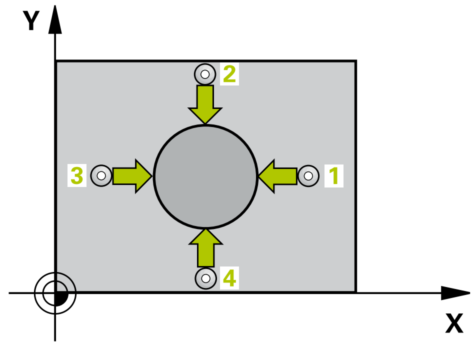
Q305=12	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (Q301=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

16.7 PRESET FROM OUTSIDE OF CIRCLE (Cycle 413, ISO: G413)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the actual values in the Q parameters listed below.
6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Please note while programming:

- The smaller the stepping angle Q247, the less accurately the control can calculate the preset. Minimum input value: 5°
- Program a stepping angle to be less than 90°; input range -120° to 120°

NOTICE

Danger of collision!

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

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

NOTICE

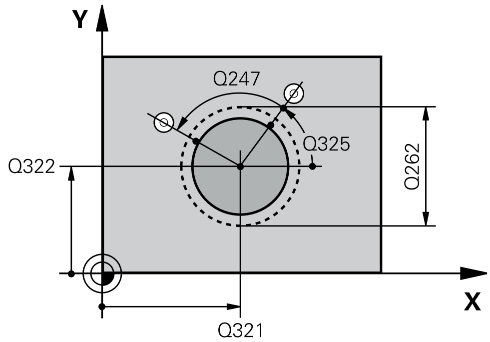
Danger of collision!

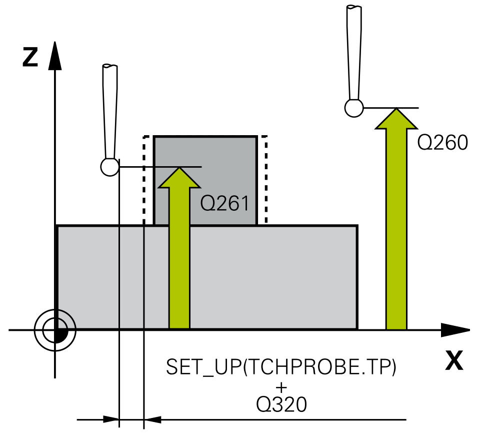
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the nominal diameter of the stud.

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

Cycle parameters

- **Q321 Center in 1st axis?** (absolute): Center of the stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q322 Center in 2nd axis?** (absolute): Center of the stud in the minor axis of the working plane. If you program Q322 = 0, the control aligns the hole center point to the positive Y axis. If you program Q322 not equal to 0, then the control aligns the hole center point to the nominal position. Input range: -99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

5 TCH PROBE 413 PRESET OUTS. CIRCLE	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=15	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

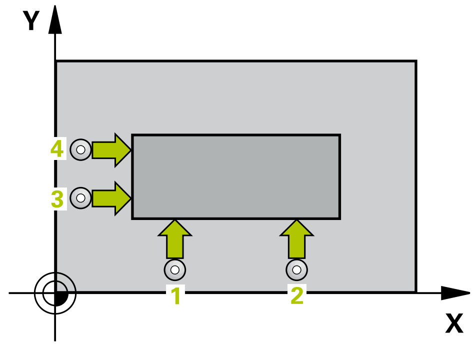
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the stud center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (Q301=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations

16.8 PRESET FROM OUTSIDE OF CORNER (Cycle 414, ISO: G414)

Cycle run

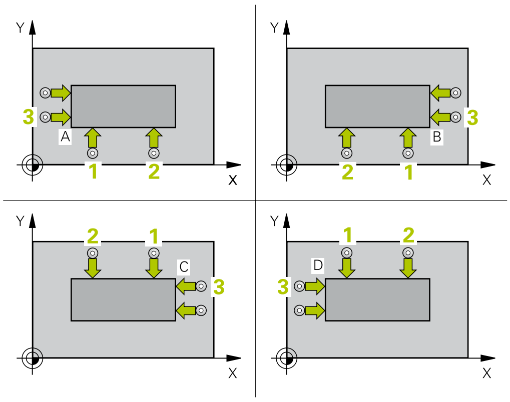
Touch probe cycle 414 finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1** (see figure at right). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the coordinates of the determined corner in the Q parameters listed below.
6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:



NOTICE

Danger of collision!

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

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

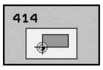
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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

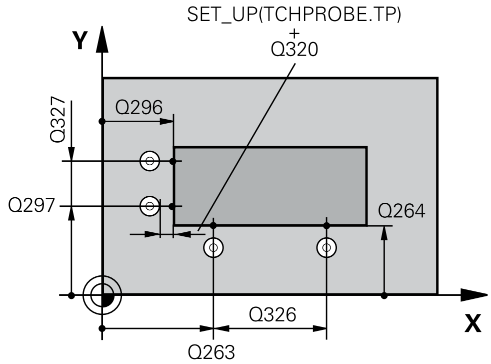
By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see figure at the right and table below).

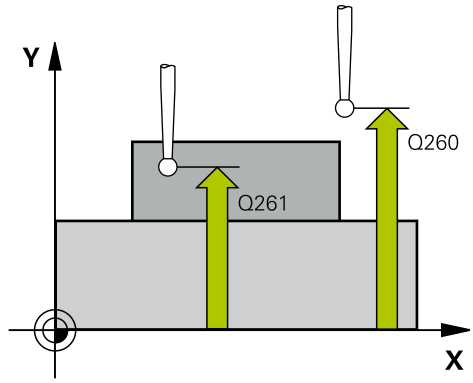
Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater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

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

5 TCH PROBE 414 PRESET INSIDE CORNER	
Q263=+37	;1ST POINT 1ST AXIS
Q264=+7	;1ST POINT 2ND AXIS
Q326=50	;SPACING IN 1ST AXIS
Q296=+95	;3RD PNT IN 1ST AXIS
Q297=+25	;3RD PNT IN 2ND AXIS
Q327=45	;SPACING IN 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q304=0	;BASIC ROTATION
Q305=7	;NUMBER IN TABLE
Q331=+0	;PRESET

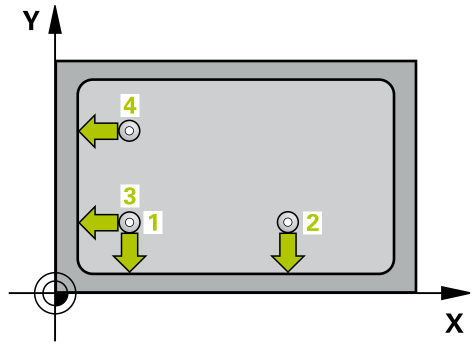
- **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:
  0: Do not execute basic rotation
  1: Execute basic rotation
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the corner coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

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

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

16.9 PRESET FROM INSIDE OF CORNER (Cycle 415, ISO: G415)

Cycle run



Touch probe cycle 415 finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1** defined in the cycle (see figure at right). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The probing direction is derived from the number by which you identify the corner.
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the coordinates of the determined corner in the Q parameters listed below.
6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Please note while programming:

NOTICE

Danger of collision!

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

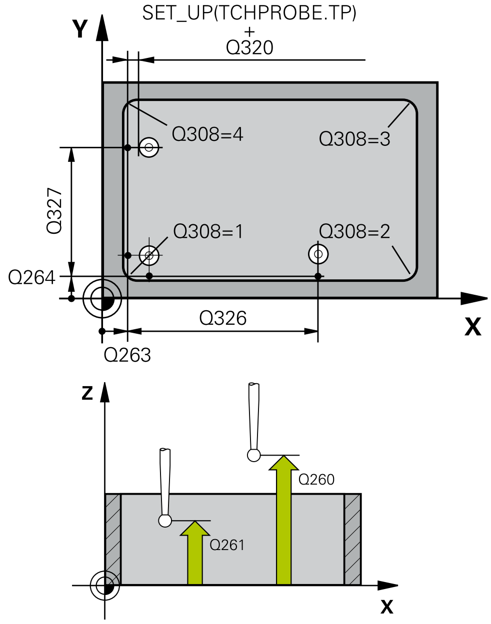
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

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

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

Cycle parameters

- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q308 Corner? (1/2/3/4)**: Number identifying the corner which the control is to set as the preset. Input range: 1 to 4
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:
  0: Do not execute basic rotation
  1: Execute basic rotation



Example

5 TCH PROBE 415 PRESET OUTS. CORNER	
Q263=+37	;1ST POINT 1ST AXIS
Q264=+7	;1ST POINT 2ND AXIS
Q326=50	;SPACING IN 1ST AXIS
Q327=45	;SPACING IN 2ND AXIS
Q308=+1	;CORNER
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q304=0	;BASIC ROTATION
Q305=7	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET

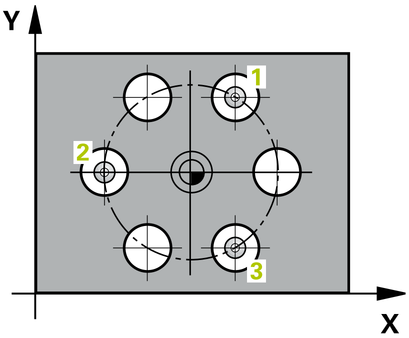
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the corner coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the corner. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

16.10 PRESET CIRCLE CENTER (Cycle 416, ISO: G416)

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 539), to the programmed center point of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
7 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

Please note while programming:

NOTICE

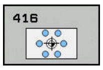
Danger of collision!

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

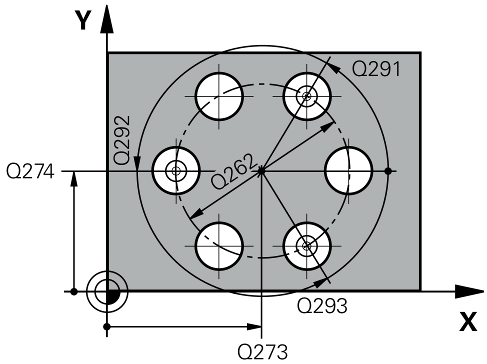
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

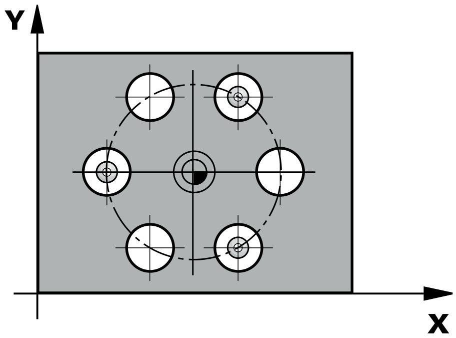
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input range: -0 to 99999.9999
- **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range: -360.0000 to 360.0000
- **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range: -360.0000 to 360.0000
- **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range: -360.0000 to 360.0000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates; input range: 0 to 9999. Depending on **Q303**, the control writes the entry to the preset table or datum table:
  If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the bolt-hole circle center. Default setting = 0. Input range: -99999.9999 to 99999.9999





Example

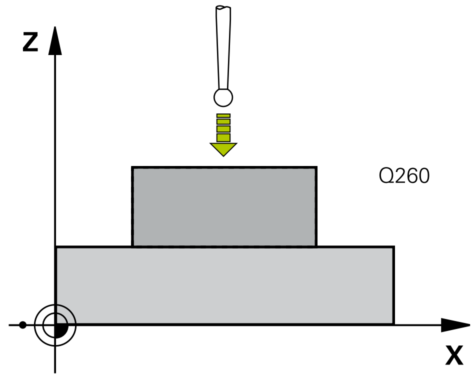
5 TCH PROBE 416 PRESET CIRCLE CENTER	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=90	;NOMINAL DIAMETER
Q291=+34	;ANGLE OF 1ST HOLE
Q292=+70	;ANGLE OF 2ND HOLE
Q293=+210	;ANGLE OF 3RD HOLE
Q261=-5	;MEASURING HEIGHT
Q260=+20	;CLEARANCE HEIGHT
Q305=12	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET
Q320=0	;SET-UP CLEARANCE

- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the bolt-hole circle center. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999

- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999

16.11 PRESET IN TOUCH PROBE AXIS (Cycle 417, ISO: G417)

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 539), to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the positive direction of the touch probe axis.
2. Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement.
3. Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters Q303 and Q305 (see "Characteristics common to all touch probe cycles for presetting", Page 592) and saves the actual values in the Q parameters listed below.

Parameter number	Meaning
Q160	Actual value of measured point

Please note while programming:

NOTICE

Danger of collision!

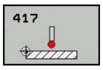
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

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

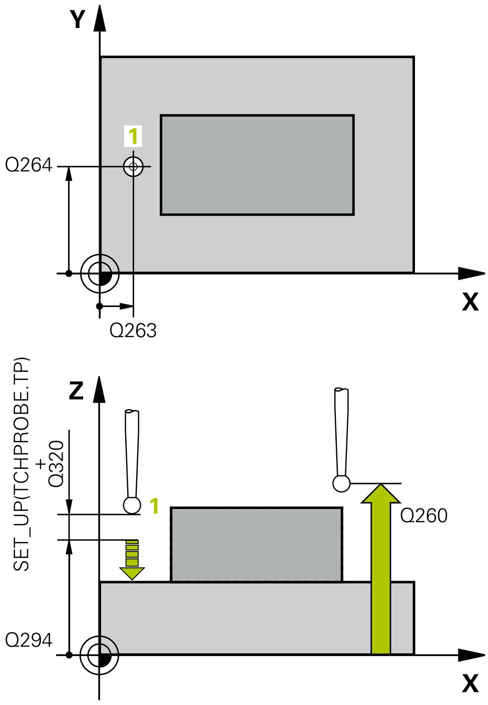
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

The control then sets the preset in this axis.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates; input range: 0 to 9999.
  If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q333 New preset in TS axis?** (absolute): Coordinate at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).



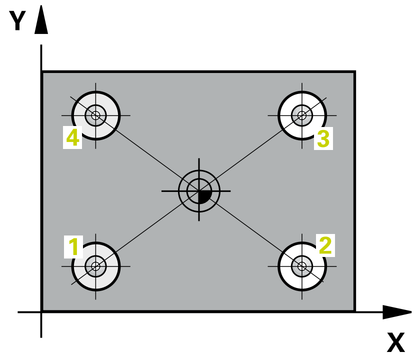
Example

5 TCH PROBE 417 PRESET IN TS AXIS	
Q263=+25	;1ST POINT 1ST AXIS
Q264=+25	;1ST POINT 2ND AXIS
Q294=+25	;1ST POINT 3RD AXIS
Q320=0	;SET-UP CLEARANCE
Q260=+50	;CLEARANCE HEIGHT
Q305=0	;NUMBER IN TABLE
Q333=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER

16.12 PRESET AT CENTER OF 4 HOLES (Cycle 418, ISO: G418)

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 539), 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 592). The control calculates the preset as the intersection of the lines connecting the centers of holes **1**/**3** and **2**/**4** and saves the actual values in the Q parameters listed below.
7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

Please note while programming:

NOTICE

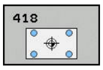
Danger of collision!

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

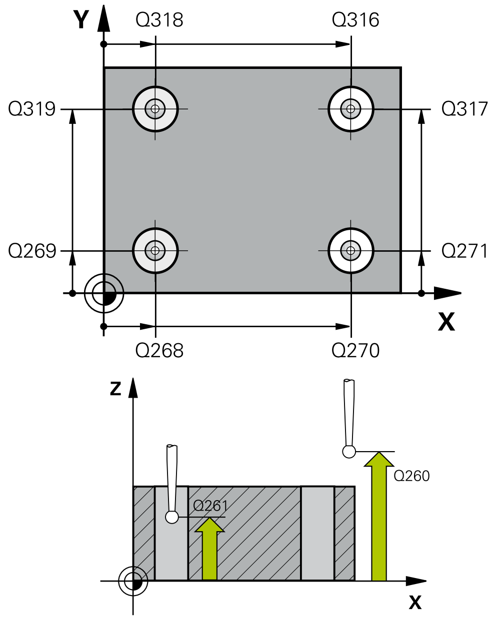
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

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

Cycle parameters



- **Q268 1st hole: center in 1st axis?** (absolute): Center of the first hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q269 1st hole: center in 2nd axis?** (absolute): Center of the first hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q270 2nd hole: center in 1st axis?** (absolute): Center of the second hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q271 2nd hole: center in 2nd axis?** (absolute): Center of the second hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q316 3rd hole: Center in 1st axis?** (absolute): Center of the third hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q317 3rd hole: Center in 2nd axis?** (absolute): Center of the third hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q318 4th hole: Center in 1st axis?** (absolute): Center of the fourth hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q319 4th hole: Center in 2nd axis?** (absolute): Center of the fourth hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999



Example

5 TCH PROBE 418 PRESET FROM 4 HOLES	
Q268=+20	;1ST CENTER 1ST AXIS
Q269=+25	;1ST CENTER 2ND AXIS
Q270=+150	;2ND CENTER 1ST AXIS
Q271=+25	;2ND CENTER 2ND AXIS
Q316=+150	;3RD CENTER 1ST AXIS
Q317=+85	;3RD CENTER 2ND AXIS
Q318=+22	;4TH CENTER 1ST AXIS
Q319=+80	;4TH CENTER 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+10	;CLEARANCE HEIGHT
Q305=12	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+0	;PRESET

- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates of the point of intersection of the connecting lines; input range: 0 to 9999.
  If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q331 New preset in reference axis?** (absolute): Coordinate in the reference axis at which the control should set the intersection of the connecting lines. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the minor axis at which the control should set the intersection of the connecting lines. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis

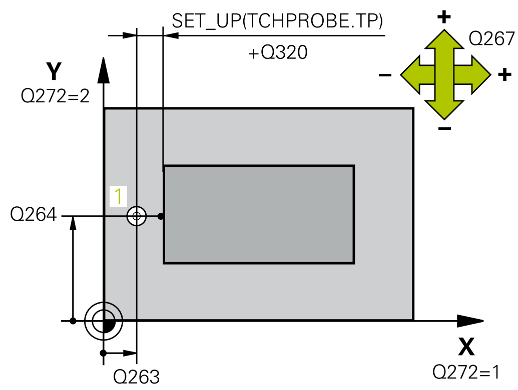
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q384 Probe TS axis: Coord. 3rd axis?** (absolute): Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if Q381 = 1. Input range: -99999.9999 to 99999.9999
- **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999

16.13 PRESET IN ONE AXIS (Cycle 419, ISO: G419)

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 539), to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite to the programmed direction of probing.
2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
3 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters Q303 and Q305. (see "Characteristics common to all touch probe cycles for presetting", Page 592)



Please note while programming:

NOTICE

Danger of collision!

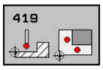
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

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

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

If you want to save the preset in several axes in the preset table, you can use Cycle 419 several times in a row. However, you also have to reactivate the preset number after every run of Cycle 419. If you work with preset 0 as active preset, this process is not required.

Cycle parameters

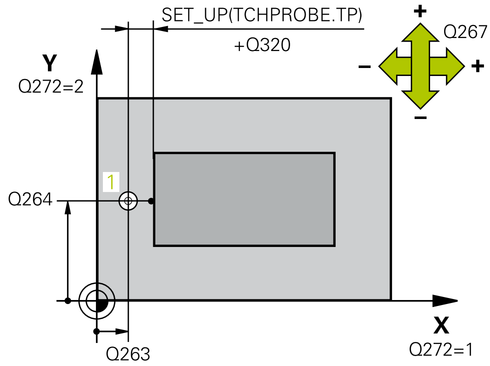


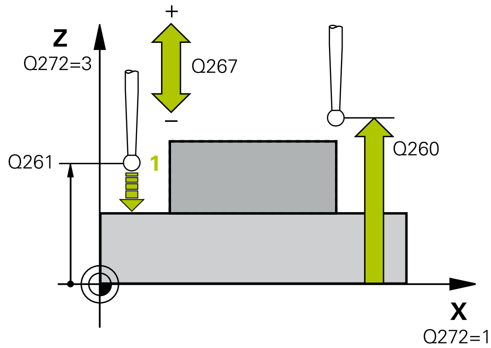
- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis

Axis assignment

Active touch probe axis: Q272 = 3	Associated reference axis: Q272 = 1	Associated minor axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction



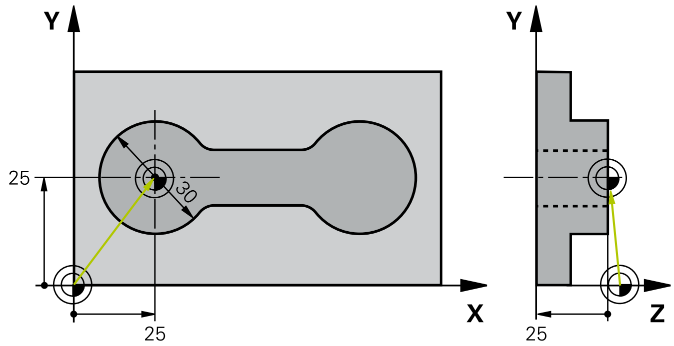


Example

```
5 TCH PROBE 419  PRESET IN ONE AXIS
    Q263=+25    ;1ST POINT 1ST AXIS
    Q264=+25    ;1ST POINT 2ND AXIS
    Q261=+25    ;MEASURING HEIGHT
    Q320=0      ;SET-UP CLEARANCE
    Q260=+50    ;CLEARANCE HEIGHT
    Q272=+1     ;MEASURING AXIS
    Q267=+1     ;TRAVERSE DIRECTION
    Q305=0      ;NUMBER IN TABLE
    Q333=+0     ;PRESET
    Q303=+1     ;MEAS. VALUE TRANSFER
```

- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates; input range: 0 to 9999.
  If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation
  If **Q303 = 0**, the control writes the data to the datum table. The datum is not automatically activated.
- **Q333 New preset in TS axis?** (absolute): Coordinate at which the control should set the preset. Default setting = 0. Input range: -99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:
  -1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 592)
  0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

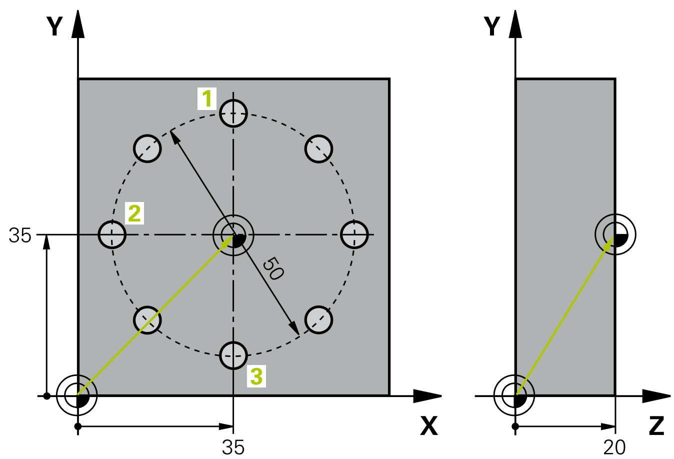
16.14 Example: Presetting in center of a circular segment and on top surface of workpiece



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!

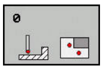
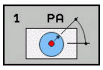
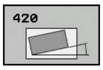
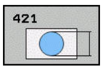
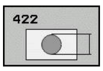
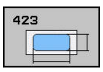
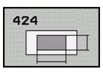
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

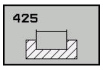
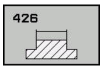
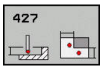
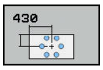
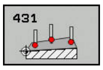
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

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

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

The control offers twelve cycles for measuring workpieces automatically.

Soft key	Cycle	Page
	0 REFERENCE PLANE Measuring a coordinate in a selectable axis	654
	1 POLAR PRESET Measuring a point in a probing direction	655
	420 MEASURE ANGLE Measuring an angle in the working plane	656
	421 MEASURE HOLE Measuring the position and diameter of a hole	659
	422 MEASURE CIRCLE OUTSIDE Measuring the position and diameter of a circular stud	664
	423 MEASURE RECTANGLE INSIDE Measuring the position, length, and width of a rectangular pocket	669
	424 MEASURE RECTANGLE OUTSIDE Measuring the position, length, and width of a rectangular stud	672

Soft key	Cycle	Page
	425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width	675
	426 MEASURE RIDGE WIDTH (2nd soft-key row) Measuring the width of a ridge	678
	427 MEASURE COORDINATE (2nd soft-key row) Measuring any coordinate in a selectable axis	681
	430 MEASURE BOLT HOLE CIRCLE (2nd soft-key row) Measuring position and diameter of a bolt hole circle	684
	431 MEASURE PLANE (2nd soft-key row) Measuring the A and B axis angles of a plane	687

Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles 0 and 1), you can have the control record the measurement results in a log. In the respective probing cycle you can define if the control is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log to a file, the control by default saves the data as an ASCII file. The control will save the file in the directory that also contains the associated NC program.

> **i** Use the HEIDENHAIN data transfer software TNCRemo if you wish to output the measuring log over the data interface.

Example: Measuring log for touch probe cycle 421:

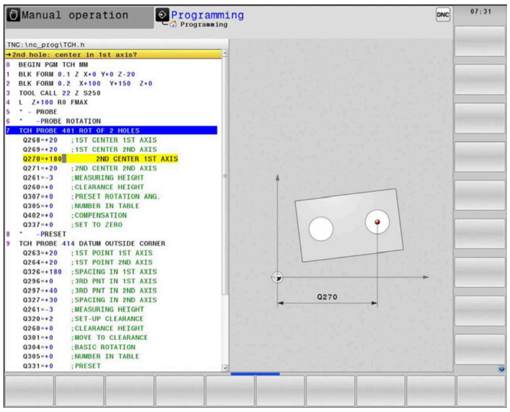
Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005
Time: 6:55:04
Measuring program: TNC:\GEH35712\CHECK1.H

Nominal values:	
Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000
Given limit values:	
Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000
Minimum limit for center in minor axis:	64.9000
Maximum dimension for hole:	12.0450
Minimum dimension for hole:	12.0000
Actual values:	
Center in reference axis:	50.0810
Center in minor axis:	64.9530
Diameter:	12.0259
Deviations:	
Center in reference axis:	0.0810
Center in minor axis:	-0.0470
Diameter:	0.0259
Further measuring results: Measuring height:	-5.0000

End of measuring log

Measurement results in Q parameters



The control saves the measurement results of the respective probing cycle in the globally effective Q parameters Q150 to Q160. Deviations from the nominal values are saved in parameters Q161 to Q166. Note the table of result parameters listed with every cycle description.

During cycle definition, the control also shows the result parameters for the respective cycle in a help graphic (see figure at upper right). The highlighted result parameter belongs to that input parameter.

Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters Q180 to Q182.

Class of results	Parameter value
Measurement results are within tolerance	Q180 = 1
Rework is required	Q181 = 1
Scrap	Q182 = 1

The control sets the rework or scrap marker as soon as one of the measuring values is out of tolerance. To determine which of the measuring results is out of tolerance, check the measuring log, or compare the respective measuring results (Q150 to Q160) with their limit values.

In Cycle 427 the control assumes by default that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.

> The control also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

With most cycles for workpiece inspection, you can have the control perform tolerance monitoring. This requires that you define the necessary limit values during cycle definition. If you do not wish to monitor for tolerances, simply leave the default value 0 for this parameter set this parameter unchanged.

Tool monitoring

With some cycles for workpiece inspection, you can have the control perform tool monitoring. The control then monitors whether

- The tool radius should be compensated because of the deviations from the nominal value (values in Q16x)
- The deviations from the nominal value (values in Q16x) are greater than the tool breakage tolerance.

Tool compensation

This function works only:

- If the tool table is active.
- If tool monitoring is switched on in the cycle: Set **Q330** unequal to 0 or enter a tool name. To enter the tool name, press the associated soft key. The control no longer displays the right single quotation mark.

If you perform several compensation measurements, the control adds the respective measured deviation to the value stored in the tool table.

Milling tool: If you refer to a milling cutter in parameter Q330, the appropriate values are compensated in the following way: The control basically always compensates the tool radius in column DR of the tool table, even if the measured deviation is within the specified tolerance. You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: rework required).

Turning tool: (only applies to Cycles 421, 422, 427) If you refer to a turning tool in parameter Q330, the appropriate values in row DZL and DXL, respectively, will be compensated. The control also monitors the breakage tolerance, which is defined in column LBREAK.You can inquire whether re-working is necessary via parameter Q181 in the NC program (Q181=1: rework required).

Tool breakage monitoring

This function works only:

- If the tool table is active.
- If tool monitoring is switched on in the cycle (set Q330 unequal to 0)
- If the breakage tolerance RBREAK for the tool number entered in the table is greater than 0

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

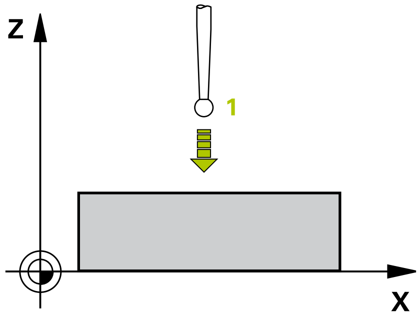
The control will output an error message and stop the program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time, the tool will be deactivated in the tool table (column TL = L).

Reference system for measurement results

The control transfers all measurement results, which reference the active coordinate system, or as the case may be, the shifted or/and rotated/tilted coordinate system, to the result parameters and the log file.

17.2 REFERENCE PLANE (Cycle 0, ISO: G55)

Cycle run



1. In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
2. Next, the touch probe performs probing at the probing feed rate (**F** column). The probing direction must be defined in the cycle.
3. After the control has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. In addition, the control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters Q115 to Q119. For the values in these parameters the control does not account for the stylus length and radius.

Please note while programming:

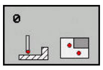
NOTICE

Danger of collision!

The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position in order to avoid a collision when the programmed pre-positioning point is approached

Cycle parameters



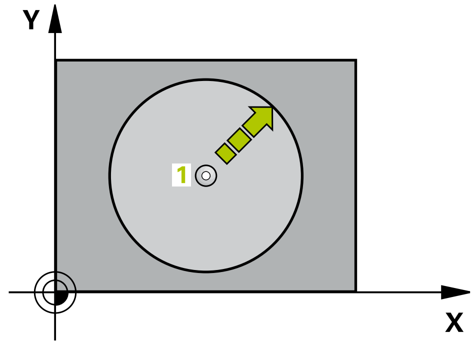
- **Parameter number for result?**: Enter the number of the Q parameter to which you want to assign the coordinate. Input range: 0 to 1999
- **Probing axis/probing direction?**: Select the probing axis with the axis key or the alphabetic keyboard, entering the algebraic sign for the probing direction. Confirm with the **ENT** key. Input range: All NC axes
- **Position value?**: Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input range: -99999.9999 to 99999.9999
- To confirm your input, press the **ENT** key.

Example

```
67 TCH PROBE 0.0  REF. PLANE Q5 X-
68 TCH PROBE 0.1  X+5 Y+0 Z-5
```

17.3 POLAR PRESET (Cycle 1)

Cycle run



Touch probe cycle 1 measures any position on the workpiece in any probing direction.

1. In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
2. Next, the touch probe performs probing at the probing feed rate (**F** column). During probing, the control moves the touch probe simultaneously in two axes (depending on the Probing angle). Specify the probing direction by entering a polar angle in the cycle.
3. After the control has saved the position, the touch probe returns to the starting point. The control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters Q115 to Q119.

Please note while programming:

NOTICE

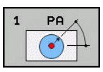
Danger of collision!

The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position in order to avoid a collision when the programmed pre-positioning point is approached

The probing axis defined in the cycle specifies the probing plane:
Probing axis X: X/Y plane
Probing axis Y: Y/Z plane
Probing axis Z: Z/X plane

Cycle parameters



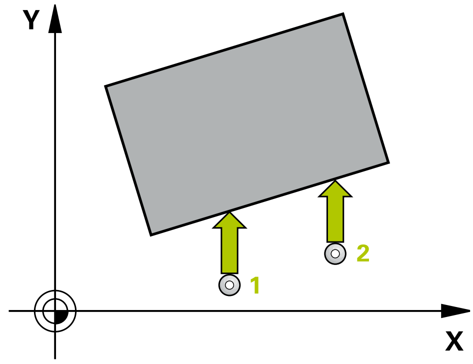
- **Probing axis?**: Specify the probing axis with the axis key or the alphabetic keyboard. Confirm with the **ENT** key. Input range: **X**, **Y** or **Z**
- **Probing angle?**: Angle, measured from the probing axis, at which the touch probe is to move. Input range: -180.0000 to 180.0000
- **Position value?**: Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input range: -99999.9999 to 99999.9999
- To confirm your input, press the **ENT** key.

Example

```
67 TCH PROBE 1.0  POLAR PRESET
68 TCH PROBE 1.1  X ANGLE: +30
69 TCH PROBE 1.2  X+5 Y+0 Z-5
```

17.4 MEASURE ANGLE (Cycle 420, ISO: G420)

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 539), to the programmed touch point **1**. The sum of Q320, **SET_UP** and the ball-tip radius is taken into account for probe movements in any probing direction. When the probe movement starts, the center of the ball tip will be offset by this sum in the direction opposite the probing direction.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

Parameter number	Meaning
Q150	The measured angle is referenced to the reference axis of the machining plane.

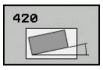
Please note while programming:

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

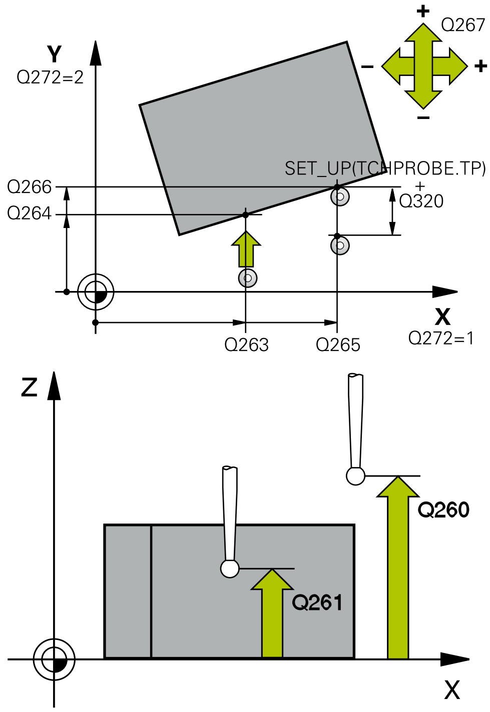
If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:

- If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
- If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of **Q320**, **SET_UP**, and the ball-tip radius, even when probing in the tool axis direction. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999



Example

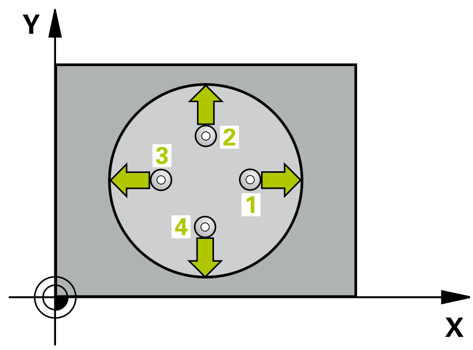
5 TCH PROBE 420 MEASURE ANGLE	
Q263=+10	;1ST POINT 1ST AXIS
Q264=+10	;1ST POINT 2ND AXIS
Q265=+15	;2ND PNT IN 1ST AXIS
Q266=+95	;2ND POINT 2ND AXIS
Q272=1	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q281=1	;MEASURING LOG

- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR420.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with **NC Start**)

17.5 MEASURE HOLE (Cycle 421, ISO: G421)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the SET_UP column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

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

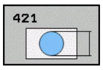
The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°

If you reference parameter Q330 to a turning tool, the following applies:

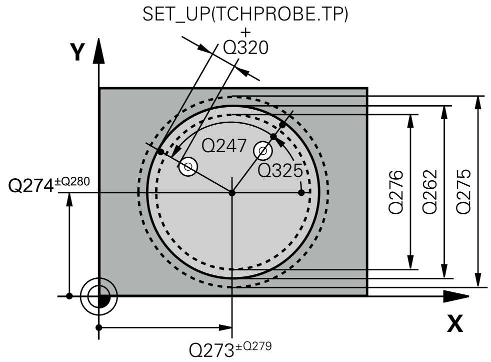
- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the control compensates the position of the turning tool, the corresponding values in rows DZL and DXL, respectively, will be compensated.
- The control also monitors the breakage tolerance, which is defined in column LBREAK.

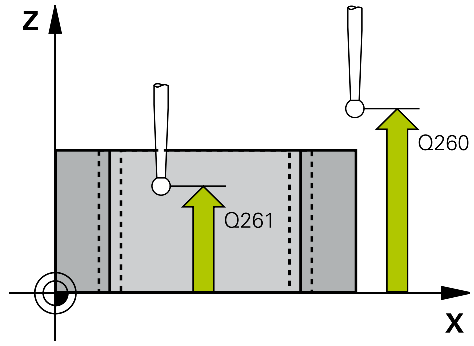
If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the hole in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the hole in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000
- **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999





Example

5 TCH PROBE 421 MEASURE HOLE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT

- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q275 Maximum limit of size for hole?**: Maximum permissible diameter for the hole (circular pocket). Input range: 0 to 99999.9999
- **Q276 Minimum limit of size?**: Minimum permissible diameter for the hole (circular pocket). Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Definition of whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR421.TXT** by default in the directory that also contains the associated NC program.
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message

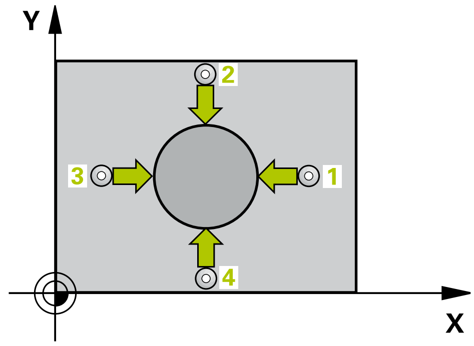
Q301=1	**;MOVE TO CLEARANCE**
Q275=75.12	**;MAXIMUM LIMIT**
Q276=74.95	**;MINIMUM LIMIT**
Q279=0.1	**;TOLERANCE 1ST CENTER**
Q280=0.1	**;TOLERANCE 2ND CENTER**
Q281=1	**;MEASURING LOG**
Q309=0	**;PGM STOP TOLERANCE**
Q330=0	**;TOOL**
Q423=4	**;NO. OF PROBE POINTS**
Q365=1	**;TYPE OF TRAVERSE**

- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 652). 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
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter Q330. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool** Q498=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle 800 and parameter **Reverse the tool** Q498=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter Q330. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle 800, **Angle of incidence?** parameter**Angle of incidence?** Q531. Input range: -180° to +180°

17.6 MEAS. CIRCLE OUTSIDE (Cycle 422, ISO: G422)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

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

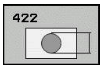
The smaller the stepping angle, the less accurately the control can calculate the dimensions of the stud. Minimum input value: 5°.

If you reference parameter Q330 to a turning tool, the following applies:

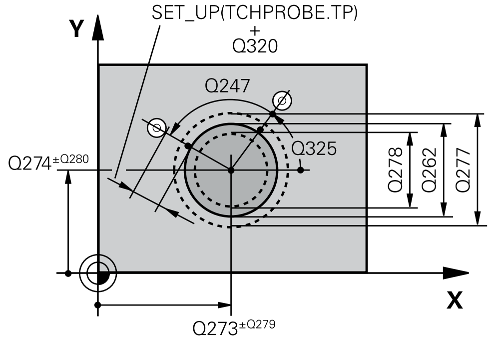
- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the control compensates the position of the turning tool, the corresponding values in rows DZL and DXL, respectively, will be compensated.
- The control also monitors the breakage tolerance, which is defined in column LBREAK.

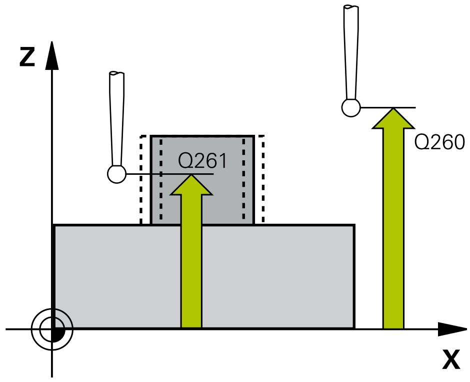
If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the diameter of the stud. Input range: 0 to 99999.9999
- **Q325 Starting angle?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: -360.000 to 360.000
- **Q247 Intermediate stepping angle?** (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the machining direction (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.0000 to 120.0000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points





Example

```
5 TCH PROBE 422  MEAS. CIRCLE
   OUTSIDE
   Q273=+50   ;CENTER IN 1ST AXIS
   Q274=+50   ;CENTER IN 2ND AXIS
   Q262=75    ;NOMINAL DIAMETER
   Q325=+90   ;STARTING ANGLE
   Q247=+30   ;STEPPING ANGLE
   Q261=-5    ;MEASURING HEIGHT
   Q320=0     ;SET-UP CLEARANCE
   Q260=+10   ;CLEARANCE HEIGHT
   Q301=0     ;MOVE TO CLEARANCE
   Q277=35.15;MAXIMUM LIMIT
   Q278=34.9  ;MINIMUM LIMIT
```

- **Q277 Maximum limit of size for stud?**: Maximum permissible diameter for the stud. Input range: 0 to 99999.9999
- **Q278 Minimum limit of size for stud?**: Minimum permissible diameter for the stud. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR422.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 652). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Tool number in the TOOL.T tool table
- **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:
  4: Use 4 measuring points (default setting)
  3: Use 3 measuring points

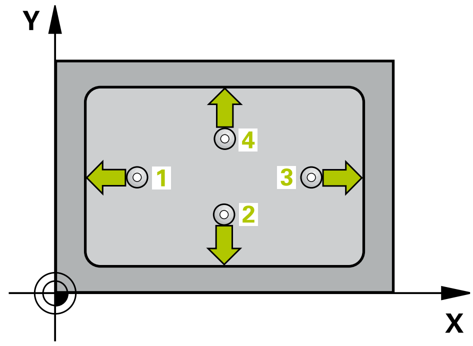
Q279=0.05	;TOLERANCE 1ST CENTER
Q280=0.05	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE

- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (Q301=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter Q330. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool** Q498=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle 800 and parameter **Reverse the tool** Q498=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter Q330. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle 800, **Angle of incidence?** parameter**Angle of incidence?** Q531. Input range: -180° to +180°

17.7 MEASURE RECTANGLE INSIDE (Cycle 423, ISO: G423)

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 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

Please note while programming:

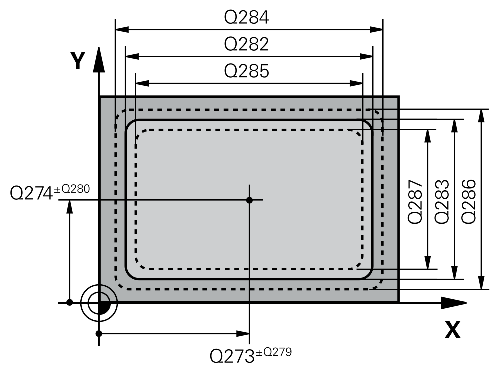
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

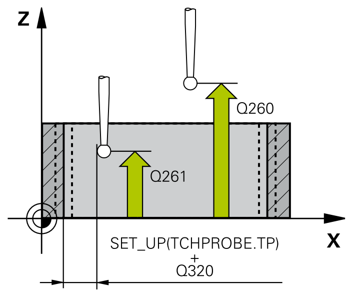
If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

Cycle parameters

423

- **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the pocket in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the pocket in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q282 1st side length (nominal value)?**: Pocket length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q283 2nd side length (nominal value)?**: Pocket length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q284 Max. size limit 1st side length?**: Maximum permissible length of the pocket. Input range: 0 to 99999.9999
- **Q285 Min. size limit 1st side length?**: Minimum permissible length of the pocket. Input range: 0 to 99999.9999
- **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the pocket. Input range: 0 to 99999.9999





Example

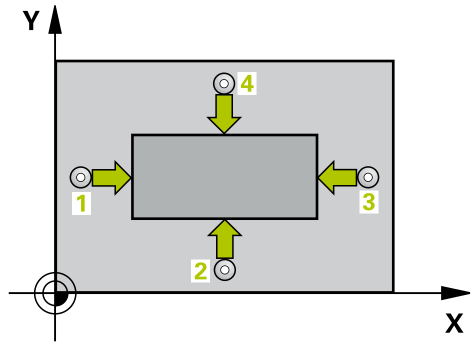
5 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50	**;CENTER IN 1ST AXIS**
Q274=+50	**;CENTER IN 2ND AXIS**
Q282=80	**;FIRST SIDE LENGTH**
Q283=60	**;2ND SIDE LENGTH**
Q261=-5	**;MEASURING HEIGHT**
Q320=0	**;SET-UP CLEARANCE**
Q260=+10	**;CLEARANCE HEIGHT**
Q301=1	**;MOVE TO CLEARANCE**
Q284=0	**;MAX. LIMIT 1ST SIDE**
Q285=0	**;MIN. LIMIT 1ST SIDE**
Q286=0	**;MAX. LIMIT 2ND SIDE**
Q287=0	**;MIN. LIMIT 2ND SIDE**
Q279=0	**;TOLERANCE 1ST CENTER**
Q280=0	**;TOLERANCE 2ND CENTER**
Q281=1	**;MEASURING LOG**

- **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the pocket. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR423.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen.Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 652). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Tool number in the TOOL.T tool table

Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

17.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, ISO: G424)

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.

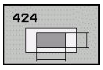
1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

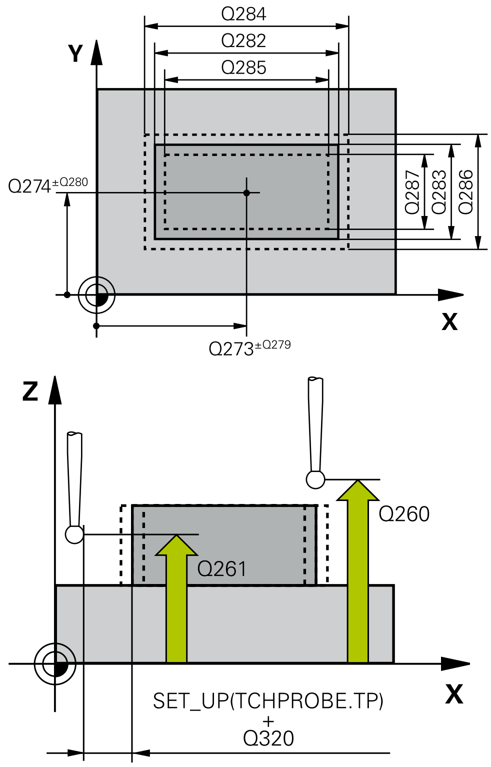
Please note while programming:

Before the cycle definition, 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 reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q282 1st side length (nominal value)?**: Stud length, parallel to the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q283 2nd side length (nominal value)?**: Stud length, parallel to the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q284 Max. size limit 1st side length?**: Maximum permissible length of the stud. Input range: 0 to 99999.9999
- **Q285 Min. size limit 1st side length?**: Minimum permissible length of the stud. Input range: 0 to 99999.9999



Example

5 TCH PROBE 424 MEAS. RECTAN. OUTS.	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;2ND CENTER 2ND AXIS
Q282=75	;FIRST SIDE LENGTH
Q283=35	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q284=75.1	;MAX. LIMIT 1ST SIDE
Q285=74.9	;MIN. LIMIT 1ST SIDE

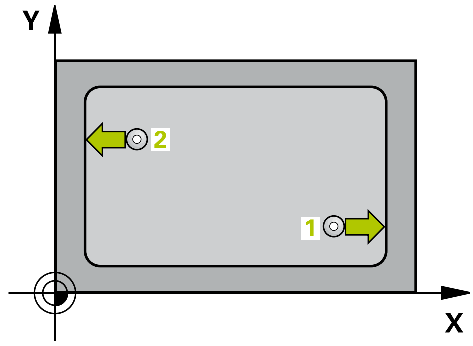
- **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the stud. Input range: 0 to 99999.9999
- **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the stud. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR424.TXT** by default in the directory that also contains the associated .h file.
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 652). 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, ISO: G425)

Cycle run

Touch probe cycle 425 measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation value in a Q parameter.



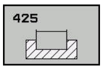
1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe table.
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the positive direction of the programmed axis.
3 If you enter an offset for the second measurement, the control then moves the touch probe (if required, at clearance height) to the next touch point **2** and probes that point. If the nominal length is large, the control moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the control measures the width in the exact opposite direction.
4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

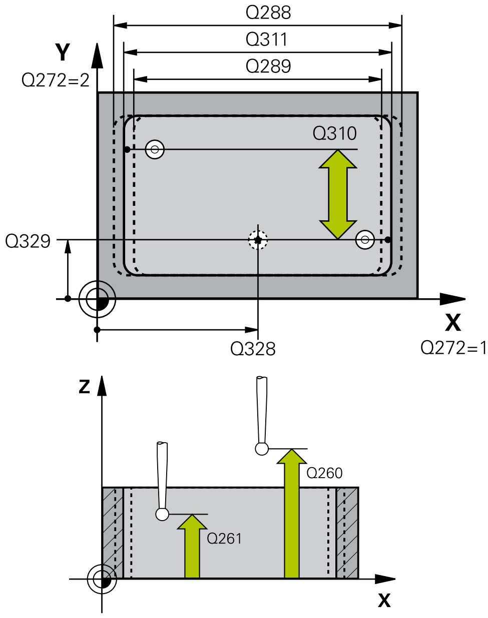
Please note while programming:

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

Cycle parameters



- **Q328 Starting point in 1st axis?** (absolute): Starting point for probing in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q329 Starting point in 2nd axis?** (absolute): Starting point for probing in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q310 Offset for 2nd measuremnt (+/-)?** (incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the control does not offset the touch probe. Input range: -99999.9999 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q311 Nominal length?** : Nominal value of the length to be measured. Input range: 0 to 99999.9999
- **Q288 Maximum limit of size?**: Maximum permissible length. Input range: 0 to 99999.9999
- **Q289 Minimum limit of size?**: Minimum permissible length. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR425.TXT** by default in the directory that also contains the associated .h file.
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.



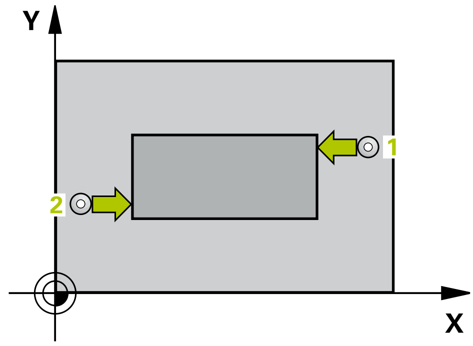
Example

5 TCH PROBE 425 MEASURE INSIDE WIDTH	
Q328=+75	**;STARTNG PNT 1ST AXIS**
Q329=-12.5	**;STARTNG PNT 2ND AXIS**
Q310=+0	**;OFFS. 2ND MEASUREMNT**
Q272=1	**;MEASURING AXIS**
Q261=-5	**;MEASURING HEIGHT**
Q260=+10	**;CLEARANCE HEIGHT**
Q311=25	**;NOMINAL LENGTH**
Q288=25.05	**;MAXIMUM LIMIT**
Q289=25	**;MINIMUM LIMIT**
Q281=1	**;MEASURING LOG**
Q309=0	**;PGM STOP TOLERANCE**
Q330=0	**;TOOL**
Q320=0	**;SET-UP CLEARANCE**
Q301=0	**;MOVE TO CLEARANCE**

- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 652). Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.
- **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points

17.10 MEASURE RIDGE WIDTH (Cycle 426, ISO: G426)

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.

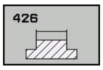
1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column 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

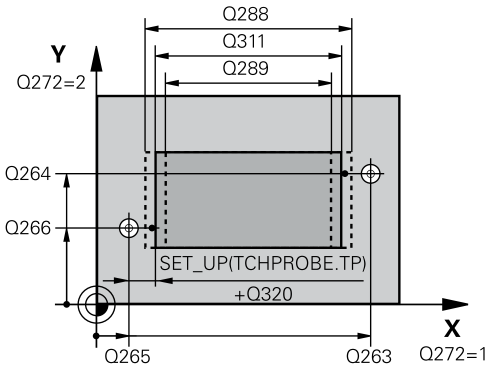
Please note while programming:

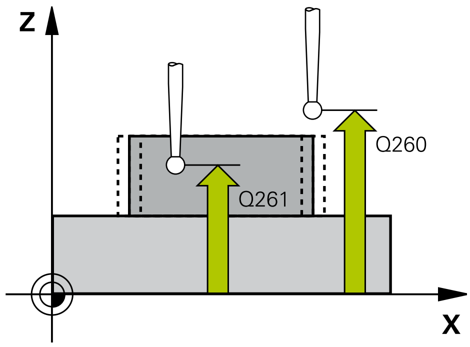
Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q311 Nominal length?** : Nominal value of the length to be measured. Input range: 0 to 99999.9999
- **Q288 Maximum limit of size?**: Maximum permissible length. Input range: 0 to 99999.9999
- **Q289 Minimum limit of size?**: Minimum permissible length. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR426.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.





Example

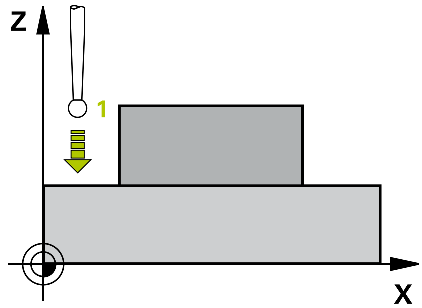
```
5 TCH PROBE 426  MEASURE RIDGE
    WIDTH
    Q263=+50    ;1ST POINT 1ST AXIS
    Q264=+25    ;1ST POINT 2ND AXIS
    Q265=+50    ;2ND PNT IN 1ST AXIS
    Q266=+85    ;2ND PNT IN 2ND AXIS
    Q272=2      ;MEASURING AXIS
    Q261=-5     ;MEASURING HEIGHT
    Q320=0      ;SET-UP CLEARANCE
    Q260=+20    ;CLEARANCE HEIGHT
    Q311=45     ;NOMINAL LENGTH
    Q288=45     ;MAXIMUM LIMIT
    Q289=44.95;MINIMUM LIMIT
    Q281=1      ;MEASURING LOG
    Q309=0      ;PGM STOP TOLERANCE
    Q330=0      ;TOOL
```

- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 652). 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, ISO: G427)

Cycle run

Touch probe cycle 427 determines a coordinate in a selectable axis and saves the value in a system parameter. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction.
2 Then the control positions the touch probe to the specified touch point **1** in the working plane and measures the actual value in the selected axis.
3 Finally, the control returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

Parameter number	Meaning
Q160	Measured coordinate

Please note while programming:

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

If an axis of the active working plane is defined as the measuring axis (Q272 = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (Q267).

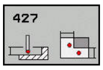
If the touch probe axis is defined as the measuring axis (Q272 = 3), the control will perform a tool length compensation.

If you reference parameter Q330 to a turning tool, the following applies:

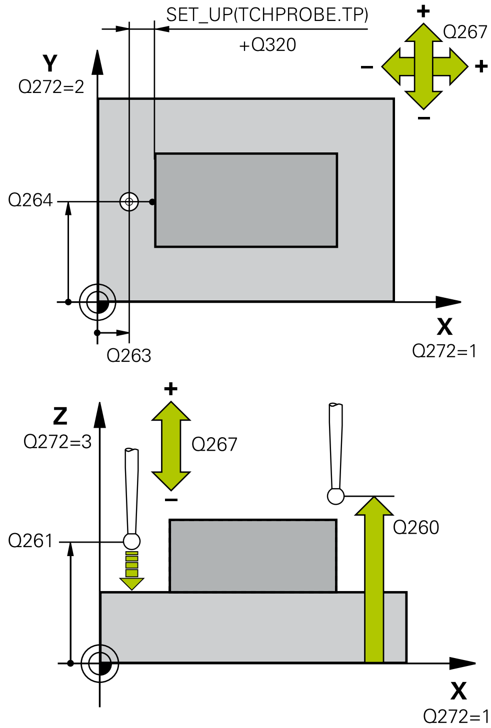
- Parameters Q498 and Q531 have to be described
- The information of the parameters Q498, Q531, for example from Cycle 800, has to match this information
- If the control compensates the position of the turning tool, the corresponding values in rows DZL and DXL, respectively, will be compensated.
- The control also monitors the breakage tolerance, which is defined in column LBREAK.

If you reference parameter Q330 to a milling tool, the information of the parameters Q498 and Q531 has no effect.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **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



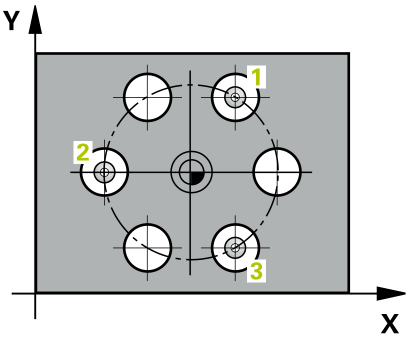
Example

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 652). 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.
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter Q330. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle 800 and parameter **Reverse the tool** Q498=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle 800 and parameter **Reverse the tool** Q498=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter Q330. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle 800, **Angle of incidence?** parameter**Angle of incidence?** Q531. Input range: -180° to +180°

17.12 MEASURE BOLT HOLE CIRCLE (Cycle 430, ISO: G430)

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.

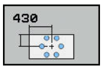
1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 539), to the programmed center point of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to 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

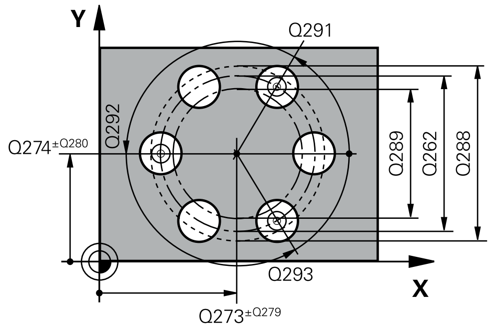
Please note while programming:

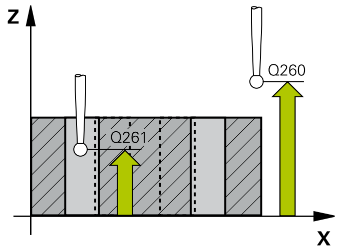
> Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
>
> Cycle 430 only monitors for tool breakage; there is no automatic tool compensation.

Cycle parameters



- **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q262 Nominal diameter?**: Enter the diameter of the hole. Input range: 0 to 99999.9999
- **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane. Input range: -360.0000 to 360.0000
- **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane. Input range: -360.0000 to 360.0000
- **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane. Input range: -360.0000 to 360.0000
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made. Input range: -99999.9999 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q288 Maximum limit of size?**: Maximum permissible diameter of bolt hole circle. Input range: 0 to 99999.9999





Example

```
5 TCH PROBE 430 MEAS. BOLT HOLE
   CIRC
   Q273=+50    ;CENTER IN 1ST AXIS
   Q274=+50    ;CENTER IN 2ND AXIS
   Q262=80     ;NOMINAL DIAMETER
   Q291=+0     ;ANGLE OF 1ST HOLE
   Q292=+90    ;ANGLE OF 2ND HOLE
   Q293=+180 ;ANGLE OF 3RD HOLE
   Q261=-5     ;MEASURING HEIGHT
   Q260=+10    ;CLEARANCE HEIGHT
   Q288=80.1 ;MAXIMUM LIMIT
```

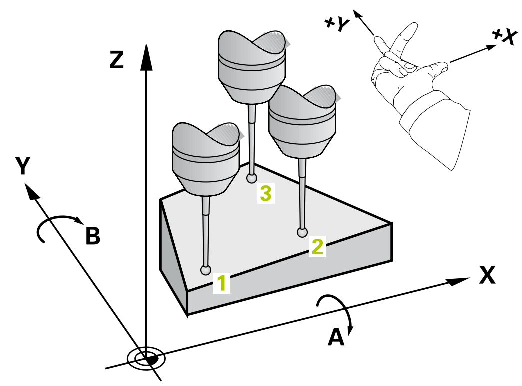
- **Q289 Minimum limit of size?**: Minimum permissible diameter of bolt hole circle. Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the reference axis of the working plane. Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the minor axis of the working plane. Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR430.TXT** in the directory that also contains the associated NC program
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 652). 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, ISO: G431)

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 539), to the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the set-up clearance in the direction opposite to the direction of probing.
2 The touch probe returns to the clearance height and then moves in the working plane to touch point **2** and measures the actual value of the second touch point in the plane.
3 The touch probe returns to the clearance height and then moves in the working plane to touch point **3** and measures the actual value of the third touch point in the plane.
4 Finally the control returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

Parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to Q175	Measured values in the touch probe axis (first to third measurement)

Please note while programming:

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

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

The spatial angles that are needed for tilting the working plane are saved in parameters Q170 – Q172. With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.

The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

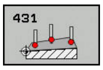
NOTICE

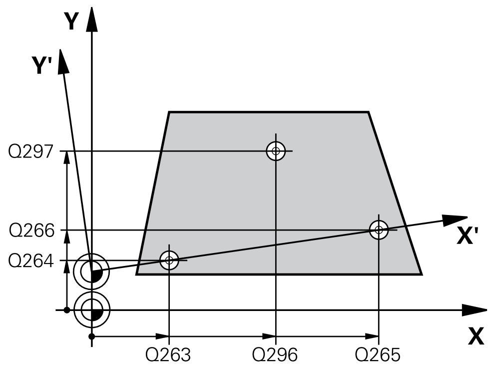
Danger of collision!

When you enter the values into the preset table and then tilt the tool by programming the spatial angles SPA=0; SPB=0; SPC=0, there are multiple solutions in which the tilting axes are at 0.

- Make sure to program SYM (SEQ) + or SYM (SEQ) -

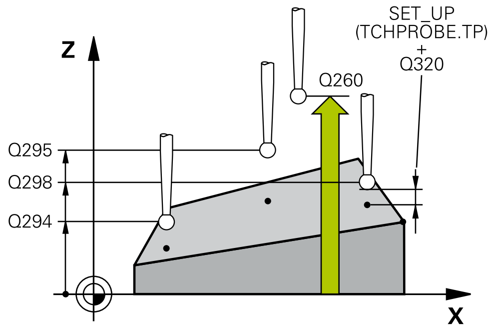
Cycle parameters





- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999

- **Q295 2nd measuring point in 3rd axis?** (absolute): Coordinate of the second touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999
- **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q298 3rd measuring point in 3rd axis?** (absolute): Coordinate of the third touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the directory that also contains the associated NC program
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.



Example

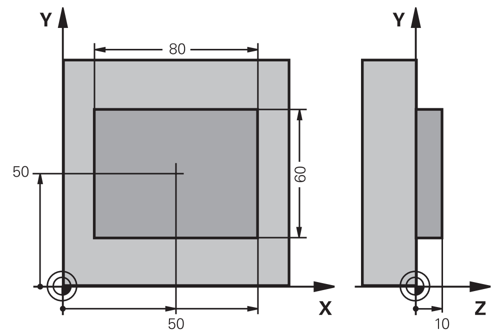
5 TCH PROBE 431 MEASURE PLANE	
Q263=+20	;1ST POINT 1ST AXIS
Q264=+20	;1ST POINT 2ND AXIS
Q294=-10	;1ST POINT 3RD AXIS
Q265=+50	;2ND PNT IN 1ST AXIS
Q266=+80	;2ND PNT IN 2ND AXIS
Q295=+0	;2ND PNT IN 3RD AXIS
Q296=+90	;3RD PNT IN 1ST AXIS
Q297=+35	;THIRD POINT 2ND AXIS
Q298=+12	;3RD PNT IN 3RD AXIS
Q320=0	;SET-UP CLEARANCE
Q260=+5	;CLEARANCE HEIGHT
Q281=1	;MEASURING LOG

17.14 Programming Examples

Example: Measuring and reworking a rectangular stud

Program run

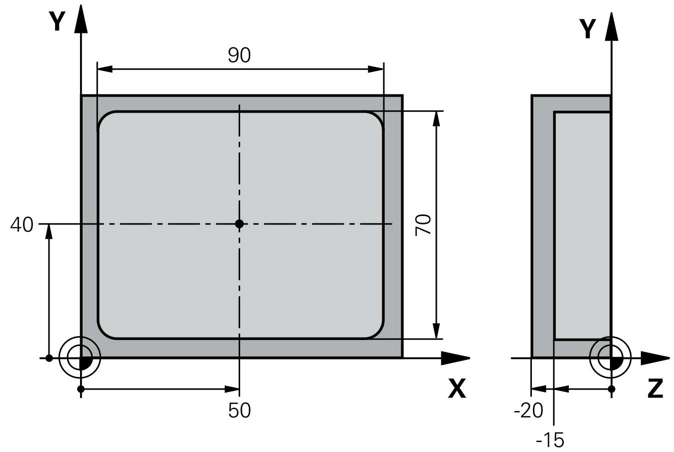
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



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 FOR MILLNG	
Q203=+10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=Q1 ;FIRST SIDE LENGTH	Length in X variable for roughing and finishing
Q219=Q2 ;2ND SIDE LENGTH	Length in Y variable for roughing and finishing
Q220=0 ;CORNER RADIUS	
Q221=0 ;ALLOWANCE IN 1ST AXS	
17 CYCL CALL M3	Cycle call
18 LBL 0	End of subprogram
19 END PGM BEAMS MM	

Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM BSMEAS MM	
1 TOOL CALL 1 Z	Tool call: touch probe
2 L Z+100 R0 FMAX	Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+40 ;CENTER IN 2ND AXIS	
Q282=90 ;FIRST SIDE LENGTH	Nominal length in X
Q283=70 ;2ND SIDE LENGTH	Nominal length in Y
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+20 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=90.15 ;MAX. LIMIT 1ST SIDE	Maximum limit in X
Q285=89.95 ;MIN. LIMIT 1ST SIDE	Minimum limit in X
Q286=70.1 ;MAX. LIMIT 2ND SIDE	Maximum limit in Y
Q287=69.9 ;MIN. LIMIT 2ND SIDE	Minimum limit in Y
Q279=0.15 ;TOLERANCE 1ST CENTER	Permissible position deviation in X
Q280=0.1 ;TOLERANCE 2ND CENTER	Permissible position deviation in Y
Q281=1 ;MEASURING LOG	Save measuring log to a file
Q309=0 ;PGM STOP TOLERANCE	Do not display an error message in case of a tolerance violation
Q330=0 ;TOOL	No tool monitoring
4 L Z+100 R0 FMAX M2	Retract the tool, end program
5 END PGM BSMEAS MM	

18

Touch Probe Cycles: Special Functions

18.1 Fundamentals

Overview

NOTICE

Danger of collision!

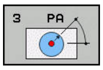
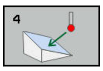
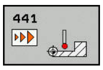
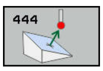
When running touch probe cycles, Cycle 400 to 499 for coordinate transformation must not be active.

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

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

HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.

The control provides cycles for the following special purposes:

Soft key	Cycle	Page
	3 **MEASURING** Measuring cycle for defining OEM cycles	695
	4 **MEASURING IN 3-D** Measuring any position	697
	441 **FAST PROBING** Measuring cycle for defining various touch probe parameters	717
	444 **PROBING IN 3-D** Measuring any position	699

18.2 MEASURE (Cycle 3)

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.

The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other measuring cycles, do not apply in Touch Probe Cycle 3.

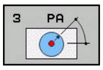
Remember that the control always writes to four successive Q parameters.

If the control was not able to determine a valid touch point, the NC program is run without error message. In this case the control assigns the value -1 to the fourth result parameter so that you can deal with the error yourself.

The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.

With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle runs through the probe input X12 or X13.

Cycle parameters



- **Parameter number for result?**: Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input range: 0 to 1999
- **Probing axis?**: Enter the axis in whose direction the probe is to move and confirm with the **ENT** key. Input range: X, Y, or Z
- **Probing angle?**: Angle, measured from the defined **probing axis** in which the touch probe is to move. Confirm with **ENT**. Input range: -180.0000 to 180.0000
- **Maximum measuring range?**: Enter the maximum traverse distance from the starting point by which the touch probe is to move. Confirm with ENT. Input range: -99999.9999 to 99999.9999
- **Feed rate measurement**: Enter the feed rate for measurement in mm/min. Input range: 0 to 3000.000
- **Maximum retraction distance?**: Traverse distance in the direction opposite the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision. Input range: 0 to 99999.9999
- **Reference system? (0=ACT/1=REF)**: Define whether the probing direction and measuring result should reference the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**):
  0: Probe in the current system and save the measuring result to the **ACT** system
  1: Probe in the machine-based REF system and save the measuring result to the REF system.
- **Error mode? (0=OFF/1=ON)**: Specify whether the control is to issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the control saves the value **-1** in the fourth result parameter and continues the cycle:
  0: Display error message
  1: Do not display error message

Example

```
4 TCH PROBE 3.0  MEASURING
5 TCH PROBE 3.1  Q1
6 TCH PROBE 3.2  X ANGLE:  +15
7 TCH PROBE 3.3  ABST +10  F100 MB1
   REFERENCE SYSTEM: 0
8 TCH PROBE 3.4  ERRORMODE1
```

18.3 MEASURING IN 3-D (Cycle 4)

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:

> The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.
>
> Ensure during pre-positioning that the control moves the probe-tip center without compensation to the defined position!
>
> Remember that the control always writes to four successive Q parameters.

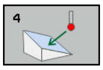
NOTICE

Danger of collision!

If the control was not able to determine a valid touch point, the 4th result parameter will have the value -1. The control does **not** interrupt the program run!

- Make sure that all touch points can be reached.

Cycle parameters



- **Parameter number for result?**: Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input range: 0 to 1999
- **Relative measuring path in X?**: X component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999
- **Relative measuring path in Y?**: Y component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999
- **Relative measuring path in Z?**: Z component of the direction vector defining the direction in which the touch probe is to move. Input range: -99999.9999 to 99999.9999
- **Maximum measuring range?**: Enter the maximum traverse distance from the starting point by which the touch probe may move along the direction vector. Input range: -99999.9999 to 99999.9999
- **Feed rate measurement**: Enter the feed rate for measurement in mm/min. Input range: 0 to 3000.000
- **Maximum retraction distance?**: Traverse distance in the direction opposite the probing direction, after the stylus was deflected. Input range: 0 to 99999.9999
- **Reference system? (0=ACT/1=REF)**: Specify whether the result of probing is to be saved in the input coordinate system (**ACT**), or with respect to the machine coordinate system (**REF**):
  0: Save measuring result to the **ACT** system
  1: Save measuring result to the **REF** system

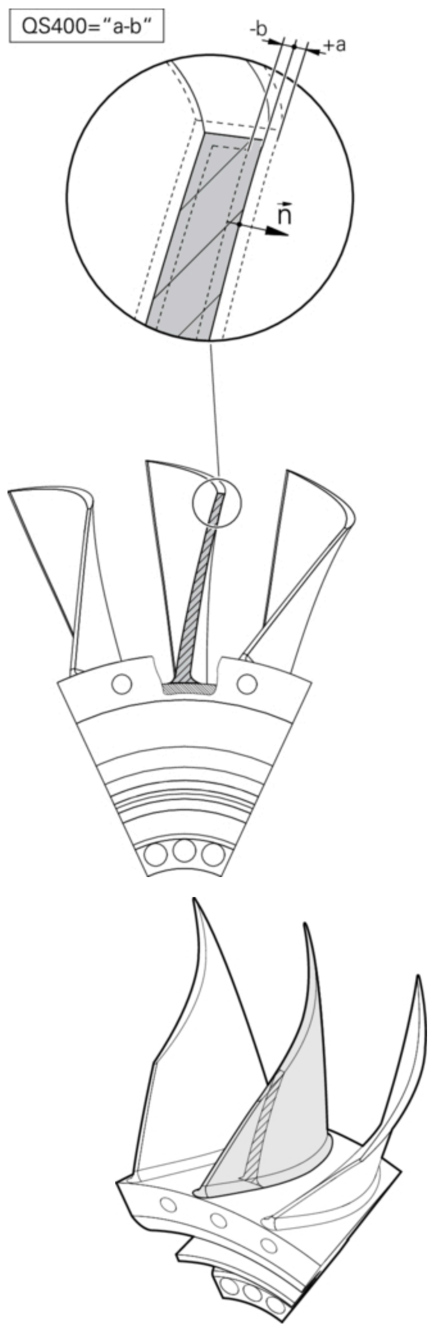
Example

```
4 TCH PROBE 4.0  MEASURING IN 3-D
5 TCH PROBE 4.1  Q1
6 TCH PROBE 4.2  IX-0.5 IY-1 IZ-1
7 TCH PROBE 4.3 ABST+45 F100 MB50
   REFERENCE SYSTEM:0
```

18.4 3D PROBING (Cycle 444)

Cycle run

Cycle 444 checks one specific point on the surface of a component. This cycle is used, for example, to measure free-form surfaces of molded parts. It can be determined whether a point on the surface of the component lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.



Cycle 444 probes any point in three dimensions, and determines the deviation to a nominal coordinate. A normal vector, defined in parameters **Q581**, **Q582**, and **Q583**, is used for this purpose. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface, and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range **QS400** defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the control outputs a log and the deviations are stored in the system parameters listed below.

Cycle run

1. Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate: Distance = ball-tip radius + **SET_UP** value from the tchprobe.tp table (TNC:\table \tchprobe.tp) + **Q320** Pre-positioning takes a clearance height into account. For more information on the probing logic, see "Executing touch probe cycles", Page 539
2. The touch probe then moves to the nominal coordinate. The probing distance is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.
3. After the control has saved the position, the touch probe is retracted and stopped. The control saves the measured coordinates of the contact point in Q parameters.
4. Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

System parameter

The control stores the probing results in the following parameters:

System parameter	Meaning
Q151	Measured position in reference axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in reference axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3-D deviation ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

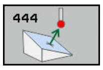
Log function

Once probing has finished, the control generates a log in HTML format. The control saves the log in the same folder in which the *.h file is located (as long as no path has been configured for FN16).

The log includes the following contents:

- Defined nominal coordinate
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")
- (If a tolerance QS400 was defined) Upper and lower allowances are output, as well as the determined deviation along the normal vector
- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the reference axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the minor axis of the working plane. Input range: -99999.9999 to 99999.9999
- **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis. Input range: -99999.9999 to 99999.9999
- **Q581 Surface-normal in ref. axis?** Enter here the surface normal in the direction of the reference axis. The surface normal of a point is normally output by a CAD/CAM system. Input range: -10 to 10
- **Q582 Surface-normal in minor axis?** Enter here the surface normal in the direction of the minor axis. The surface normal of a point is normally output by a CAD/CAM system. Input range: -10 to 10
- **Q583 Surface-normal in tool axis?** Enter here the surface normal in the direction of the tool axis. The surface normal of a point is normally output by a CAD/CAM system. Input range: -10 to 10
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur. Input range: -99999.9999 to 99999.9999

Example

```
4 TCH PROBE 444 PROBING IN 3-D
    Q263=+0     ;1ST POINT 1ST AXIS
    Q264=+0     ;1ST POINT 2ND AXIS
    Q294=+0     ;1ST POINT 3RD AXIS
    Q581=+1     ;NORMAL IN REF. AXIS
    Q582=+0     ;NORMAL IN MINOR AXIS
    Q583=+0     ;NORMAL IN TOOL AXIS
    Q320=+0     ;SAFETY CLEARANCE
    Q260=100    ;CLEARANCE HEIGHT
    QS400="1-1";TOLERANCE
    Q309=+0     ;ERROR REACTION
```

- **QS400 Tolerance value?** Specify a tolerance range that will be monitored by the cycle. The tolerance defines the deviation permitted along the surface normals. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. (The surface normal is defined by Q581 - Q583, and the nominal coordinate is defined by Q263, Q264, and Q294.) The tolerance value is distributed over the axes, depending on the normal vector:
  Example: QS400 ="0.4-0.1" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate -0.1. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate - 0.1".
  Example: QS400 ="0.4" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate".
  Example: QS400 ="-0.1" means: upper allowance = nominal coordinate, lower allowance = nominal coordinate -0.1. The following tolerance range thus results for the cycle: "nominal coordinate" to "nominal coordinate -0.1".
  Example: QS400 =" " means: No tolerance band.
  Example: QS400 ="0" means: No tolerance band.
  Example: QS400 ="0.1+0.1" means: No tolerance band.
- **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 the interrupt 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 value of the determined actual coordinate along the surface normal vector is less than the nominal coordinate, the control displays a message and interrupts program run. An undersize has occurred. On the other hand, there is no error reaction if the value determined along the surface normal vector is greater than the nominal coordinate.

Please note while programming!

In order to obtain exact results from the touch probe being used, you should perform a 3-D calibration before executing Cycle 444. Software option 92, 3D-ToolComp, is required for 3-D calibration.

Cycle 444 generates a measuring log in HTML format.

An error message is output if a mirroring (Cycle 8) or scaling (Cycle 11, 26) is active before Cycle 444 is run.

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.

For probing, an active TCPM will be taken into account. While the TCPM is active, probing of positions is even possible if the position resulting from the Tilt Working Plane function is inconsistent with the current position of the rotary axes.

If your machine is equipped with a 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.

Cycle 444 references all coordinates to the input system.

The control writes the measured values to return parameters, see "Cycle run", Page 699.

The workpiece status good/rework/scrap is set via Q parameter Q^83, independent of parameter Q309 (see "Cycle run", Page 699).

18.5 Calibrating a touch trigger probe

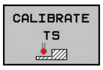
In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.

> Always calibrate a touch probe in the following cases:
>
> - Commissioning
> - Broken stylus
> - Stylus replacement
> - Change in the probe feed rate
> - Irregularities caused, for example, when the machine heats up
> - Change of active tool axis
>
> The control assumes the calibration values for the active probe system directly after the calibration process.
> The updated tool data is immediately effective. It is not necessary to repeat the tool call.

During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

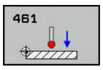
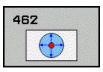
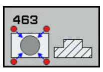
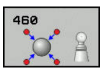
The control provides calibration cycles for calibrating the length and the radius:

- Press the **TOUCH PROBE** soft key

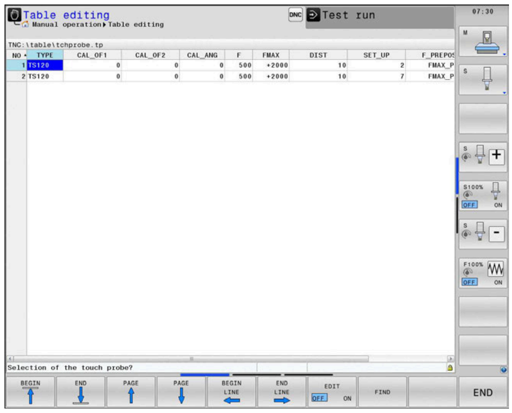


- Display the calibration cycles: Press the **CALIBRATE TS** soft key.
- Select the calibration cycle

Calibration cycles provided by the control

Soft key	Function	Page
	Calibrating the length	710
	Measure the radius and the center offset using a calibration ring	712
	Measuring the radius and the center offset using a stud or a calibration pin	714
	Measure the radius and the center offset using a calibration sphere	706

18.6 Displaying calibration values



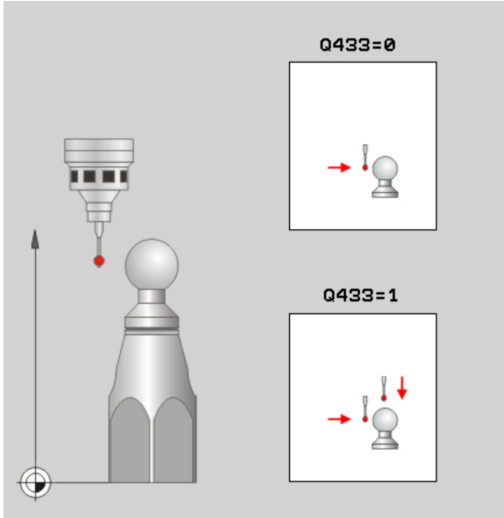
The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset values in columns **CAL_OF1** (reference axis) and **CAL_OF2** (minor axis) of the touch probe table. You can display the values on the screen by pressing the TOUCH PROBE TABLE soft key.

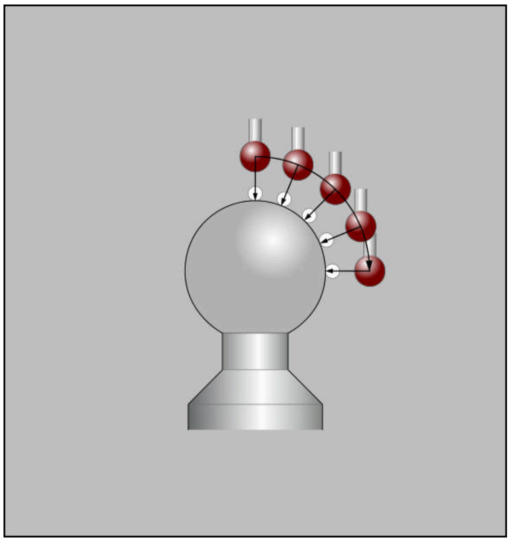
A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs. When running a touch probe cycle in the Manual Operation mode, the control saves the measuring log under the name TCHPRMAN.html. This file is stored in the folder TNC:\ *.

Ensure that the tool number of the tool table and the touch-probe number of the touch-probe table are correct. This is regardless of whether you want to use a touch-probe cycle in automatic mode or **Manual operation** mode.

For more information, see Chapter Touch-probe table

18.7 CALIBRATE TS (Cycle 460, ISO: G460)





Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration sphere. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere.

With Cycle 460 you can calibrate a triggering 3-D touch probe automatically using an exact calibration sphere.

It is also possible to capture 3-D calibration data. Software option 92, 3D-ToolComp, is required for this purpose. 3-D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3-D calibration data is stored under TNC:\system \CAL_TS<T_no.>_<T_idx.>.3DTC . The DR2TABLE column of the tool table references the 3DTC table. The 3-D calibration data is then taken into account when probing. This 3-D calibration is necessary if you want to achieve a very high accuracy with Cycle 444 3-D Probing (see "3D PROBING (Cycle 444)", Page 699).

Cycle run

The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

Radius calibration Q433=0

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3 The control first moves in the plane, depending on the reference angle (Q380).
4 The control then positions the touch probe in touch-probe axis.
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
6 Once the equator has been determined, the radius calibration begins
7 Finally, the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

Radius and length calibration Q433=1

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
3 The control first moves in the plane, depending on the reference angle (Q380).
4 The control then positions the touch probe in touch-probe axis.
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
6 Once the equator has been determined, the radius calibration begins
7 The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.
8 The control determines the length of the touch probe at the north pole of the calibration sphere.
9 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

The setting in parameter **Q455** specifies whether you can perform an additional 3-D calibration.

3-D calibration Q455= 1...30

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 After calibration of the radius and length, the control retracts the touch probe in touch-probe axis. Then the control positions the touch probe above the north pole.
3 The probing process goes from the north pole to the equator in several steps. Deviations from the nominal value, and therefore the specific deflection behavior, are thus determined
4 You can specify the number of probing points between the north pole and the equator. This number depends on input parameter Q455. A value between 1 and 30 can be programmed. If you program Q455=0, no 3-D calibration will be performed.
5 The deviations determined during the calibration are stored in a 3DTC table.
6 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

Please note while programming:

NOTICE

Danger of collision!

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

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

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

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

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

Pre-position the touch probe so that it is located approximately above the center of the calibration sphere.

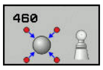
If you program Q455=0, the control will not perform a 3-D calibration.

If you program Q455=1- 30, the control will perform a 3-D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles. If you use Cycle 444, you should first perform a 3-D calibration.

If you program Q455=1- 30, a table will be stored under TNC:\Table\CAL_TS<T_no.>_<T_Idx.>.3DTC. <T_no> is the number of the touch probe, and <Idx> is its index.

If there is already a reference to a calibration table (entry in DR2TABLE), this table will be overwritten.

If there is no reference to a calibration table (entry in DR2TABLE), then, in dependence of the tool number, a reference and the associated table will be created.



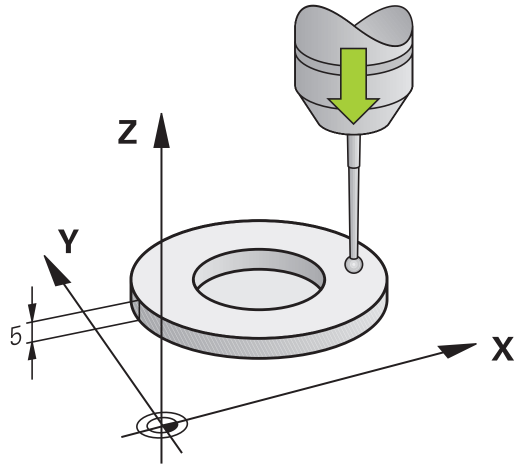
- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. Q320 is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. Input range: 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- **Q380 Ref. angle in ref. axis?** (absolute) Enter the Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range: 0 to 360.0000
- **Q433 Calibrate length (0/1)?**: Define whether the control is to calibrate the touch probe length after radius calibration, as well:
  0: Do not calibrate touch probe length
  1: Calibrate touch probe length
- **Q434 Preset for length?** (absolute): Coordinate of the calibration sphere center. This value must only be defined required if length calibration is to be carried out. Input range: -99999.9999 to 99999.9999
- **Q455 No. of points for 3-D calibrtn.?** Enter the number of touch points for 3-D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3-D calibration. During 3-D calibration, the deflecting behavior of the touch probe is determined under various angles, and the values are stored in a table. 3D-ToolComp is required for 3-D calibration. Input range: 1 to 30

Example

5 TCH PROBE 460 CALIBRATION OF TS ON A SPHERE	
Q407=12.5	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE
Q423=4	;NO. OF PROBE POINTS
Q380=+0	;REFERENCE ANGLE
Q433=0	;CALIBRATE LENGTH
Q434=-2.5	;PRESET
Q455=15	;NO. POINTS 3-D CAL.

18.8 CALIBRATE TS LENGTH (Cycle 461, ISO: G461)

Cycle run



Before starting the calibration cycle, you must set the preset in the spindle axis so that Z=0 on the machine table; you must also pre-position the touch probe above the calibration ring.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

1 The control orients the touch probe to the angle **CAL_ANG** specified in the touch probe table (only if your touch probe can be oriented).
2 The control probes from the current position in the negative spindle axis direction at the probing feed rate (column **F** from the touch probe table).
3 The control then retracts the touch probe at rapid traverse (column **FMAX** from the touch probe table) to the starting position.

Please note while programming:

NOTICE

Danger of collision!

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

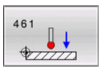
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**,Cycle **8 MIRROR IMAGE**, Cycle **10 ROTATION**,Cycles **11 SCALING**, and **26 AXIS-SPECIFIC SCALING**
- Reset any coordinate transformations beforehand

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

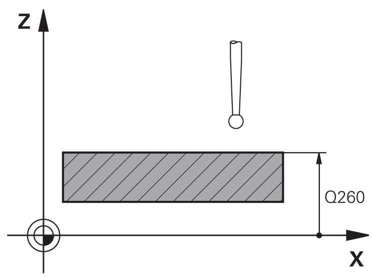
The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

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

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.



- **Q434 Preset for length?** (absolute): Preset for the length (e.g. height of the calibration ring). Input range: -99999.9999 to 99999.9999



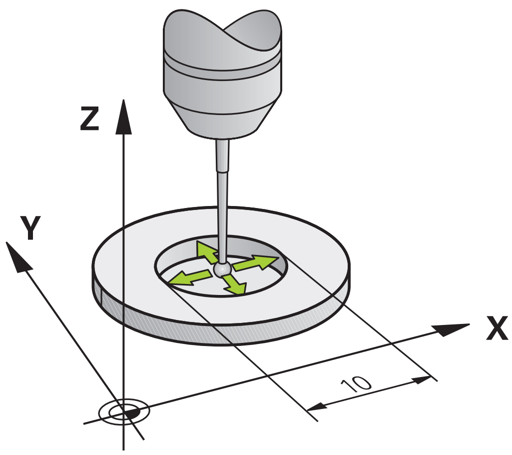
Example

```
5 TCH PROBE 461  TS CALIBRATION OF
   TOOL LENGTH
   Q434=+5      ;PRESET
```

18.9 CALIBRATE TS RADIUS INSIDE (Cycle 462, ISO: G462)

Cycle run

Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.



When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180° and then executes four more probing operations. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Please note while programming:

NOTICE

Danger of collision!

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

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

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

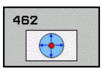
The center offset can be determined only with a suitable touch probe.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

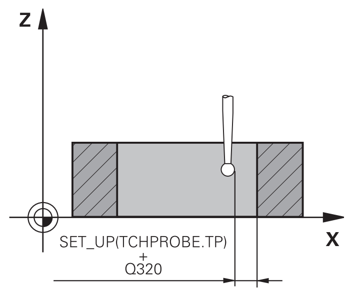
In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer. Please refer to the machine manual.

The orientation property of your touch probe is already predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

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



- **Q407 RING RADIUS** Enter the radius of the calibration ring. Input range 0 to 9.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- **Q380 Ref. angle in ref. axis?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: 0 to 360.0000



Example

```
5 TCH PROBE 462  CALIBRATION OF A
  TS IN A RING
  Q407=+5      ;RING RADIUS
  Q320=+0      ;SET-UP CLEARANCE
  Q423=+8      ;NO. OF PROBE POINTS
  Q380=+0      ;REFERENCE ANGLE
```

18.10 CALIBRATE TS RADIUS OUTSIDE (Cycle 463, ISO: G463)

Cycle run

Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration pin. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration pin.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180° and then executes four more probing operations. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Please note while programming:

NOTICE

Danger of collision!

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

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

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

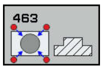
The center offset can be determined only with a suitable touch probe.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

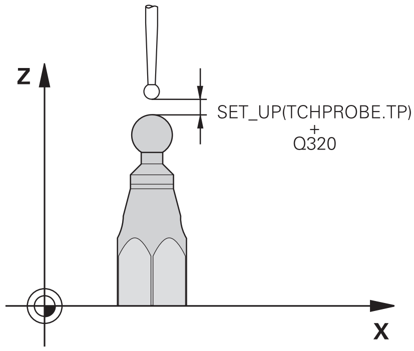
In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer. Please refer to the machine manual.

The orientation property of your touch probe is already predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

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



- **Q407 Radius of calibr. stud?**: Diameter of the calibration ring or pin. Input range: 0 to 99.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move to measuring height between measuring points
  1: Move to clearance height between measuring points
- **Q423 Number of probes?** (absolute): Number of measuring points on the diameter. Input range: 3 to 8
- **Q380 Ref. angle in ref. axis?** (absolute): Angle between the reference axis of the working plane and the first touch point. Input range: 0 to 360.0000



Example

5 TCH PROBE 463 TS CALIBRATION ON STUD	
Q407=+5	;STUD RADIUS
Q320=+0	;SET-UP CLEARANCE
Q301=+1	;MOVE TO CLEARANCE
Q423=+8	;NO. OF PROBE POINTS
Q380=+0	;REFERENCE ANGLE

18.11 FAST PROBING (Cycle 441, ISO G441)

Cycle run

You can use the touch probe cycle 441 to globally specify various touch probe parameters (e.g. the positioning feed rate) for all subsequently used touch probe cycles.

Please note while programming:

Cycle 441 sets parameters for touch probe cycles. This cycle does not perform machine movements

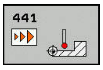
END PGM, **M2**, **M30** resets the global settings of Cycle 441.

Cycle parameter **Q399** depends on your machine configuration. Orienting the touch probe system via the NC program must be enabled by your machine tool builder.

The feed rate can also be limited by your machine tool builder. The absolute maximum feed rate is defined in the machine parameter **maxTouchFeed** (No. 122602).

Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate, also with Q397=1, only with the potentiometer for feed motions.

Cycle parameters



- **Q396 Positioning feed rate?**: Define the feed rate at which the touch probe will be moved to the specified positions. Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- **Q397 Pre-pos. at machine's rapid?**: Define whether the control, when pre-positioning the touch probe, traverses at **FMAX** feed rate (machine's rapid traverse):
  0: Pre-position with feed rate from **Q396**
  1: Pre-position with machine's rapid traverse **FMAX**Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate, also with Q397=1, only with the potentiometer for feed motions.
  The feed rate can also be limited by your machine tool builder. The absolute maximum feed rate is defined in the machine parameter **maxTouchFeed** (No. 122602).
- **Q399 Angle tracking (0/1)?**: Define whether the control orients the touch probe before each probing operation:
  0: No orientation
  1: Orient spindle before each probing operation (improves the accuracy)
- **Q400 Automatic interruption?** Define whether the control interrupts the program run and outputs the measuring results on the screen following a measuring cycle for automatic workpiece measurement:
  0: Do not interrupt program run even if, in the specific probing cycle, the output of measuring results on the screen is selected
  1: Interrupt program run and output measuring results on the screen. You can then resume the NC program run with **NC Start**.

Example

5 TCH PROBE 441 FAST PROBING	
Q 396=3000	;POSITIONING FEED RATE
Q 397=0	;SELECT FEED RATE
Q 399=1	;ANGLE TRACKING
Q 400=1	;INTERRUPTION

19

Visual Setup Control VSC (Software Option 136)

19.1 Camera-based monitoring of the setup situation VSC (option 136)

Fundamentals

In order to use the Visual Setup Control you require the following components:

- Software: Option 136 Visual Setup Control (VSC)
- Hardware: Camera system of HEIDENHAIN

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine tool builder.

Camera-based checking of the setup situation (option 136 Visual Setup Control) allows you to monitor the current setup situation before and during processing, and to compare this with a safe target status. After setup, simple cycles for automatic monitoring are available.

Reference images of the current working space are recorded in the camera system. With Cycles 600 **GLOBAL WORKING SPACE** or 601 **LOCAL WORKING SPACE**, the control produces an image of the working space and compares the image with previously prepared reference images. These cycles can highlight irregularities in the working space. The operator decides whether the NC program is interrupted in the event of an error or continues to run.

Using VSC offers the following advantages:

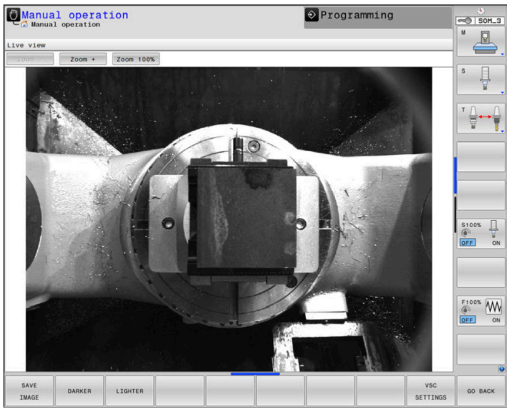
- The control can recognize elements (e.g. tools, fixtures, etc.) that are in the working space once the program has started
- If you always want to clamp a workpiece at the same position (e.g. hole at top right), the control can check the clamping situation
- For documentation purposes you can generate an image of the current workspace (e.g. of a clamping situation that is rarely used)

Terms

The following terms are used in connection with VSC:

Term	Explanation
Reference image	A reference image shows a situation in the working space that you regard as safe. Therefore only create reference images showing safe, non-hazardous situations.
Mid-value image	The control creates a mid-value image taking into account all reference images. The control compares new images with the mid-value image as part of evaluation.
Images with error	If you generate an image showing a poor situation (such as an incorrectly clamped workpiece), you can generate an "image of error" It is not advisable to highlight an error image as a reference image.
Monitoring area	Denotes an area that you highlight with the mouse. When evaluating new images, the control only refers to this area. Parts of images outside the monitoring area have no effect on the results of the monitoring process. Several monitoring areas can be defined. Monitoring areas are not linked to images.
Error	Area on an image containing a deviation from the desired position. Errors always refer to the image for which they were saved (image of error) or to the image most recently evaluated.
Monitoring phase	No further reference images are produced in the monitoring phase. You can use the cycle for automatic monitoring of your working space. In this phase, the control only issues a warning if it finds a deviation when comparing images.

Producing a live image



In the **Manual operation** mode, you can display and save the current camera view as a live image.

The control does not use the image captured here for automatic checking of the clamping situation. Images produced in this menu may be used for documentation and traceability. For example, you could record the current setup situation. The control saves the image produced as a .png file in a directory of your choosing.

Procedure

Proceed as follows to save the camera's live image:

- Press the **CAMERA** soft key

- Press the **LIVE IMAGE** soft key
- > The control displays the current camera view.
- > The control opens a pop-up window.
- Enter the desired file name
- Select the desired target directory

- Press the **OK** soft key
- > The control saves the current live image.
- Alternative: Press the **Save** button

Options in Live Image mode

The control provides the following options:

Soft key	Function
LIGHTER	Increase camera brightness The settings made here only affect Live Image mode. They have no influence on pictures taken in automatic mode.
DARKER	Reduce camera brightness The settings made here only affect Live Image mode. They have no influence on pictures taken in automatic mode.
VSC SETTINGS	Configure the field of view of the camera Refer to your machine manual. These settings can only be made after entering a code number.
GO BACK	Go back to the previous screen

Managing monitoring data

In the **Manual operation** mode you can manage images from Cycles 600 and 601.

Proceed as follows to enter the monitoring data:

- Press the **CAMERA** soft key

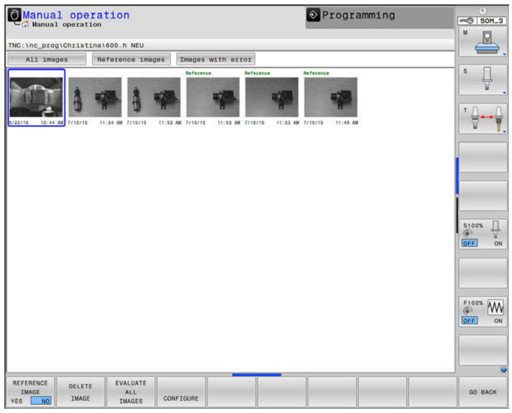
- Press the **MONITORING DATA MANAGEMENT** soft key
- > The control shows a list of the NC programs being monitored.

- Press the **OPEN** soft key
- > The control shows a list of monitoring points.
- Edit the desired data

Selecting data

You can select the buttons with the mouse. These interfaces make it easier to search and show results in a manageable way.

- **All images**: Display all images for this monitoring file
- **Reference images**: Only display reference images
- **Images with error**: Display all images where you have highlighted an error



Features of the monitoring data management

Soft key	Function
REFERENCE IMAGE YES NO	Mark selected image as a reference image Please note: A reference image shows a situation in the working space that you regard as safe. All reference images are taken into account for the evaluation. If you add or remove an image as a reference image, this has an effect on the results of image evaluation.
DELETE IMAGE	Delete image currently selected
EVALUATE ALL IMAGES	Carry out automatic image evaluation The control carries out an image evaluation according to the reference images and the monitoring areas.
CONFIGURE	Change monitoring area or highlight an error
GO BACK	Go back to the previous screen If you change the configuration, the control carries out an image evaluation.

Overview

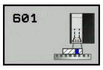
The control provides two cycles you can use for visual setup control in **Programming** mode of operation:

TOUCH PROBE

- The soft-key row shows all available touch probe functions divided into groups.

MONITORING WITH CAMERA

- Press the **MONITORING WITH CAMERA** soft key.

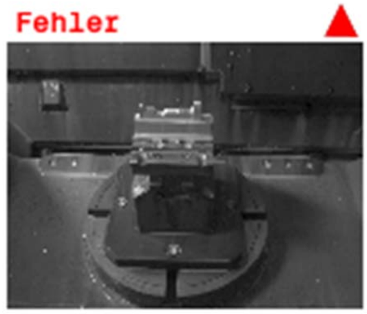
Soft key	Cycle	Page
	600 **GLOBAL WORKING SPACE**	732
	601 **LOCAL WORKING SPACE**	737

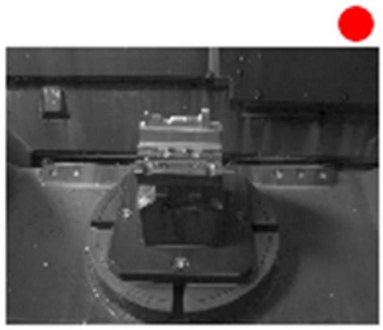
Result of the image evaluation

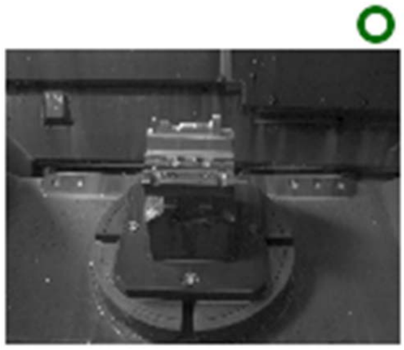
The result of the image evaluation depends on the monitoring area and the reference images. When evaluating all images, each image is evaluated according to the current configuration and the results are compared with the data last saved.

If you change the monitoring area or add/delete reference images, the images may be tagged with the following symbol:

- **Triangle:** You changed the monitoring area or the sensitivity. This has an impact on all of your reference images and the mean image, respectively. As a result of your change to the configuration, the control can no longer detect errors that had been saved for this image! The system has become less sensitive. If you would like to proceed, confirm the reduced sensitivity of the system. The new settings will be applied.
- **Solid circle:** You changed the monitoring area or the sensitivity. This has an impact on all of your reference images and the mean image, respectively. As a result of your change to the configuration, the control can now detect errors that it was unable to detect before. The system has become more sensitive. If you would like to proceed, confirm the increased sensitivity of the system. The new settings will be applied.
- **Empty circle:** No error message: All deviations saved in the image were detected. The system, therefore, has basically kept its previous sensitivity.







Configuration

You can change your settings regarding the monitoring area and error area at any time. When you press the **CONFIGURE** soft key, the soft key display changes and you can change your settings.

CONFIGURE

- You can now edit your previously entered settings. If you make a change in this menu, the results of the image evaluation may change. The same monitoring area applies to all reference images.
  Further information: "Result of the image evaluation", Page 727

DRAW AREA

- You can click on the image and draw up a rectangular frame. This way, you define a new monitoring area. (For more information, see "Fundamentals", Page 720.) If you define monitoring areas in an environment that is not always exposed in the same manner or in which differences in contrast are to be expected, false alerts will be displayed. If you draw a new monitoring area or change or delete an existing monitoring area, this will affect the result of the image evaluation. Due to the changed settings the control must check whether these changes may have an impact on the previously generated images.

DRAW ERROR

- You can click on the image and draw up a rectangular frame. This way you define a new area with errors. The area is marked red. It is recommended to mark only errors that may reoccur exactly at this spot. It is not advisable to mark an area that is contaminated with chips or drilling fluid as an error area. The errors need to be reproducible in the exact same way. (For more information, see "Fundamentals", Page 720.) If you define monitoring areas in an environment that is not always exposed in the same manner or in which differences in contrast are to be expected, false alerts will be displayed. If you draw a new error area or change or delete an existing error area, this will affect the result of the image evaluation. Due to the changed settings the control must check whether these changes may have an impact on the previously generated images. You can also draw several error areas. It does not make sense to indicate errors on reference images.

EVALUATE IMAGE

- The control checks whether and how the new settings affect this image.
  Further information: "Result of the image evaluation", Page 727

EVALUATE ALL IMAGES

- The control checks whether or how the new settings affect all images.
  Further information: "Result of the image evaluation", Page 727

SHOW AREAS

- The control shows all drawn monitoring areas.

SHOW COMPARISON

- The control compares the current image with the mean image.

SAVE AND GO BACK

- Save the current image and return to the previous screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 727

GO BACK

- You discard all changes and return to the previously displayed screen.

Defining the monitoring area

A monitoring area can be defined in the **Program run, single block** or **Program run, full sequence** operating mode. The control will prompt you to define a monitoring area. The control will display this prompt on the screen after you have started the cycle for the first time in the **Program run, single block** or **Program run, full sequence** operating mode.

A monitoring area consists of one or more windows that you draw up with your mouse. The control will only take these areas of the image into account. Errors outside the monitoring area will not be detected. The monitoring area is not linked to the images, but only to the monitoring file specified in QS600. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Monitoring areas may overlap.

Definition of the monitoring area:

1 Click the image with the mouse and draw up a new area
2 If you would like to define more than one window, press the **DRAW AREA** soft key, and repeat this process at the appropriate location.

After you have defined the monitoring area you press the following soft key, for example:

SPEICHERN UND ZURÜCK

- Save current image and return to previous screen.

Message displayed: **Monitoring point is configured: Select the soft key.**

The status display at the top right of the image will display information about the minimum number of reference images, the current number of reference images and the current number of error images.

Possible queries

The VSC cycles enter a value in parameter Q601.

The following values are possible:

- Q601 = 1: No error
- Q601 = 2: Error
- Q601 = 3: You have yet not defined a monitoring area or you did not save enough reference images
- Q601 = 10: Internal error (no signal, faulty camera, etc.)

You can use parameter Q601 for internal queries.

Further information: If-then decisions: User's Manual Conversational Programming

The following is a programming example for a query:

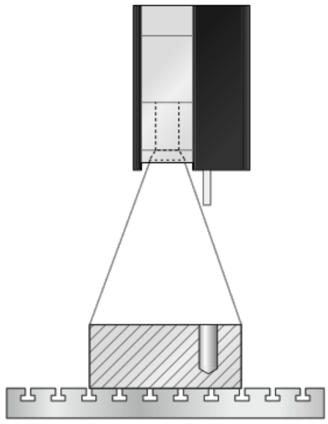
0 BEGIN PGM 5MM	
1 BLK FORM CYLINDER Z R42 L150	Blank form Cylinder
2 FUNCTION MODE MILL	Activate milling mode
3 TCH PROBE 601 LOCAL WORKING SPACE	Define Cycle 600
QS600 = OS ;MONITORING POINT	
Q309=+0 ;PGM STOP TOLERANCE	
Q613 = +0 ;KEEP CAMERA OPEN	
Q617 = 10 ;REFERENCE IMAGES	
4 FN 9: IF Q601 EQU 1 GOTO LBL 20	If parameter Q601 = 1, then jump to LBL 20
5 FN 9: IF Q601 EQU 2 GOTO LBL 21	If parameter Q601 = 2, then jump to LBL 21
6 FN 9: IF Q601 EQU 3 GOTO LBL 22	If parameter Q601 = 3, then jump to LBL 22
7 FN 9: IF Q601 EQU 10 GOTO LBL 23	If parameter Q601 = 10, then jump to LBL 23
8 TOOL CALL "GEAR_HOB_D75"	Call the tool
9 L X+... Y+... R0 FMAX	Program the machining
...	
...	
...	
57 LBL 21	Definition LBL 21
58 STOP	Program stop, the operator can check the condition of the workspace
59 LBL 0	
60 END PGM 5MM	

19.2 Workspace Global (Cycle 600)

Application

With Cycle 600, Workspace Global, you monitor the workspace of your machine tool. The control will generate an image of the current workspace from a position determined by your machine tool builder. Then, the control will match the image against previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle 600 becomes effective as soon as it has been defined, and does not have to be called. Before you can use camera monitoring, you must first generate reference images and define a monitoring area.

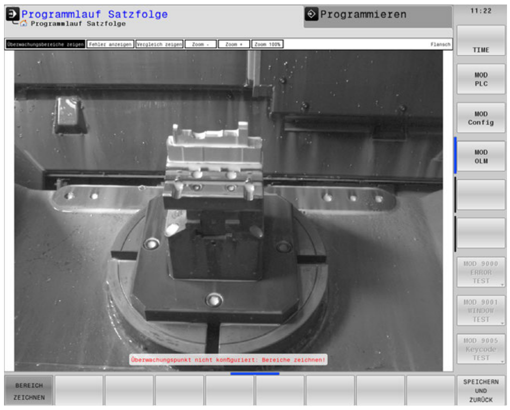
Further information: "Creating reference images", Page 733
Further information: "Monitoring phase", Page 734



Creating reference images

The control will start generating reference images as soon as you run the cycle for the first time in the Program Run, Single Block or Program Run, Full Sequence operating mode.

The following cycle run applies as long as the control has not stored sufficient reference images. Specify the number of reference images with parameter Q617.



Cycle run

1 The camera will be mounted by the machine tool builder onto the main spindle. The main spindle moves to a position defined by the machine tool builder.
2 After this position has been reached, the control will automatically open the camera cap.
3 The control will generate an image of the current condition and display it on the screen.
4 The first time this cycle is run, the following message appears at the bottom of the screen: **Monitoring point not configured: Draw areas!**
5 Define the monitoring area.
   Further information: "Defining the monitoring area", Page 730
6 You can decide whether the current image should be saved as a reference image or an error image, but you can also change the monitoring area.
   Further information: "Configuration", Page 728
7 Press the **GO BACK** soft key.
8 The control closes the camera cap.
9 Press **NC start** and run your program as usual.

After you have defined the monitoring area, the following soft keys are available in addition to the **GO BACK** soft key:

SAVE AND GO BACK

- Save current image and return to previous screen.

Message displayed: **Monitoring point is configured: Select the soft key.**

The status display at the top right of the image will display information about the minimum number of reference images, the current number of reference images and the current number of error images.

WIEDER-HOLEN

- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 727

REFERENZ-BILD

- The status display at the top right displays the word "Reference". You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the **IMAGE OF ERROR** soft key turns gray.

FEHLER-BILD

- The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an error image can never be a reference image at the same time, the **REFERENCE IMAGES** soft key turns gray.

KONFIGU-RIEREN

- The soft key row changes. You can now change your previously entered settings with respect to the monitoring area and the sensitivity. If you make a change in this menu, this may affect all of your images.
  Further information: "Configuration", Page 728

ZURÜCK

- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 727

As soon as the control has generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected, the following message is displayed: **Too few reference images: Select the next action by soft key!**. This message disappears once the number of reference images defined in parameter Q617 has been reached.

The control will generate a mean image, taking into account all reference images. During evaluation, new images are compared with the mean image, taking the variance into account. Only once all reference images are available, does the cycle no longer stop due to missing reference images.

Monitoring phase

Cycle run: Monitoring phase

1. The camera will be mounted by the machine tool builder onto the main spindle. The main spindle moves to a position defined by the machine tool builder.
2. After this position has been reached, the control will automatically open the camera cap.
3. The control will generate an image of the current condition.
4. The control will then compare the images using the mean image and the variance image.
   Further information: "Fundamentals", Page 720
5. Depending on whether the control detects an "error" (deviation), the control can now force a program cancellation (for further information, see "Fundamentals", Page 720). If parameter Q309=1 is set, the control will display the image on the screen upon detection of an error. If parameter Q309=0 is set, no image will be displayed on the screen and the program will not be stopped.
6. Finally, the control closes the camera cap.

Please note while programming:

In addition to the attribute Reference image, you can also assign the attribute Error image to your images. This assignment may impact the image evaluation.

Remember the following:

- A reference image may never be an error image at the same time.

If you change the monitoring area, this has an impact on all images.

- Ideally, you define the monitoring area only once at the beginning and then make no or only little changes to it.

The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.

- Specify a sensible number of reference images in parameter Q617. (Approx. value: 10 images).
- You can also create more reference images than you have specified in Q617.

Your machine must be prepared for camera-based monitoring!

NOTICE

Danger of collision!

Danger of contaminating the camera due to open camera lid with parameter Q613. This could lead to blurred images, the camera may be damaged.

- Close the camera lid before continuing with the process.

NOTICE

Danger of collision!

Danger of collision with automatic positioning of the camera. The camera and your machine may be damaged.

- Ask your machine manufacturer to tell you the point that the camera pre-positions to. Your machine tool builder specifies the coordinates which Cycle 600 positions to.

Cycle parameters

- **QS600** (string parameter) **Name of monitoring point?**: Enter the name of your monitoring file.
- **Q616 Feed rate for positioning?**: Feed rate the control uses to position the camera. Here, control moves the camera to a position defined by the machine tool builder.
- **Q309 PGM stop if tolerance exceeded?**: (0/1) Specify whether the control stops the program if an error is detected.
  0: The NC program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter Q601 is written to even if Q309=0.
  1: The NC program will stop after detecting an error, the generated image will be displayed in the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the control has generated enough reference images. If an error is detected, the control will display a message.
- **Q617 Number of reference images?**: Number of reference images that are required by the control for monitoring.

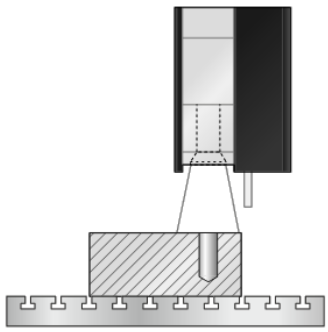
Example

4 TCH PROBE 600 GLOBAL WORKING SPACE	
QS600="OS"	;MONITORING POINT
Q616=500	;FEED RATE FOR POSITIONING
Q309=1	;PGM STOP TOLERANCE
Q617=10	;REFERENCE IMAGES

19.3 Workspace Local (Cycle 601)

Application

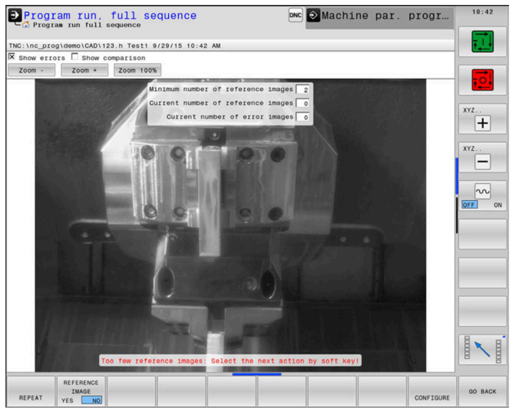
With Cycle 601, Workspace Local, you monitor the workspace of your machine tool. The control will generate an image of the current workspace from the position of the spindle at the moment the cycle is called. Then, the control will match the image against previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle 601 becomes effective as soon as it has been defined, and does not have to be called. Before you can use camera monitoring, you must first generate reference images and define a monitoring area.
Further information: "Creating reference images", Page 737
Further information: "Monitoring phase", Page 739



Creating reference images

The control will start generating reference images as soon as you run the cycle for the first time in the Program Run, Single Block or Program Run, Full Sequence operating mode.

The following cycle run applies as long as the control has not stored sufficient reference images. Specify the number of reference images with parameter Q617.



Cycle run

1 The camera will be mounted by the machine tool builder onto the main spindle.
2 The control automatically opens the camera cap.
3 The control will generate an image of the current condition and display it on the screen.
4 The first time this cycle is run, the following message appears at the bottom of the screen: "**Monitoring point not configured: Draw areas!**"
5 Define the monitoring area.
Further information: "Defining the monitoring area", Page 730
6 You can decide whether the current image should be saved as a reference image or an error image, but you can also change the monitoring area.
Further information: "Configuration", Page 728
7 Press the **GO BACK** soft key.
8 Finally, the control closes the camera cap.
9 Press **NC Start** and run your NC program as usual.

After you have defined the monitoring area, you can press the following soft keys in addition to the **GO BACK** soft key:

REPEAT

- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 727

REFERENCE IMAGE

- The status display at the top right displays the word "Reference". You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the soft key **IMAGE OF ERROR** turns gray. (For more information, see "Fundamentals", Page 720)

IMAGE OF ERROR

- The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an error image can never be a reference image at the same time, the soft key **REFERENCE IMAGE** (For more information, see "Fundamentals", Page 720)

CONFIGURE

- The soft key row changes. You are now able to change your previously entered settings with respect to the monitoring area and the sensitivity. If you carry out the change in this menu, this may impact all of your images. (For more information, see "Configuration", Page 728)

GO BACK

- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 727)

> As soon as the control has generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected, the following message is displayed: **Too few reference images: Select the next action by soft key!**. This message disappears once the number of reference images defined in parameter Q617 has been reached.

> The control will generate a mean image, taking into account all reference images. During evaluation, new images are compared with the mean image, taking the variance into account. Only once all reference images are available, does the cycle no longer stop due to missing reference images.

Monitoring phase

The monitoring phase starts as soon as the control has generated enough reference images.

Cycle run: Monitoring phase

1. The camera will be mounted by the machine tool builder onto the main spindle.
2. The control automatically opens the camera cap.
3. The control will generate an image of the current condition.
4. The control will then compare the images using the mean image and the variance image.
5. Depending on whether the control detects an "error" (deviation), the control can now force a program cancellation. If parameter Q309=1 is set, the control will display the image on the screen upon detection of an error. If parameter Q309=0 is set, no image will be displayed on the screen and the program will not be stopped.
6. Depending on the setting of parameter Q613, the control either leaves the camera cap open or closes it.

Please note while programming!

In addition to the Reference Image attribute, you can assign the Error Image attribute to your images. This assignment may impact image evaluation.

Remember the following:

- A reference image may never be an error image at the same time.

If you change the monitoring area, this has an impact on all images.

- Ideally, you define the monitoring area only once, i.e. at the beginning. and then make no or only little changes to it.

The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.

- Specify a sensible number of reference images in parameter Q617. (Approx. value: 10 images)
- You can also create more reference images than you have specified in Q617.

Your machine must be prepared for camera-based monitoring!

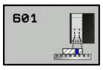
NOTICE

Danger of camera contamination due to camera cap kept open with parameter Q613.

This could result in blurred images, or the camera may be damaged.

Close the camera cap before continuing the machining process!

Cycle parameters



- **QS600** (string parameter) **Name of monitoring point?**: Enter the name of your monitoring file.
- **Q309 PGM stop if tolerance exceeded?**: (0/1) Specify whether the control stops the program if an error is detected.
  0: The NC program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter Q601 is written to even if Q309=0.
  1: The NC program will stop after detecting an error, the generated image will be displayed in the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the control has generated enough reference images. If an error is detected, the control will display a message.
- **Q613 Keep camera cap open?**: (0/1) Specify whether the control should keep the camera cap open after monitoring.
  0: The control closes the camera cap after running Cycle 601.
  1: The control keeps the camera cap open after running Cycle 601. This function is only recommended if you would like to generate yet another image of the workspace in a different position after the first cycle call of Cycle 601. In order to do so, program the new position in a linear block and then call up Cycle 601 with a new monitoring point. Program Q613=0 before continuing with the cutting operation!
- **Q617 Number of reference images?**: Number of reference images that are required by the control for monitoring.

Example

4 TCH PROBE 601 LOCAL WORKING SPACE
QS600="OS" ;MONITORING POINT
Q309=+1 ;PGM STOP TOLERANCE
Q613=0 ;KEEP CAMERA OPEN
Q617=10 ;REFERENCE IMAGES

20

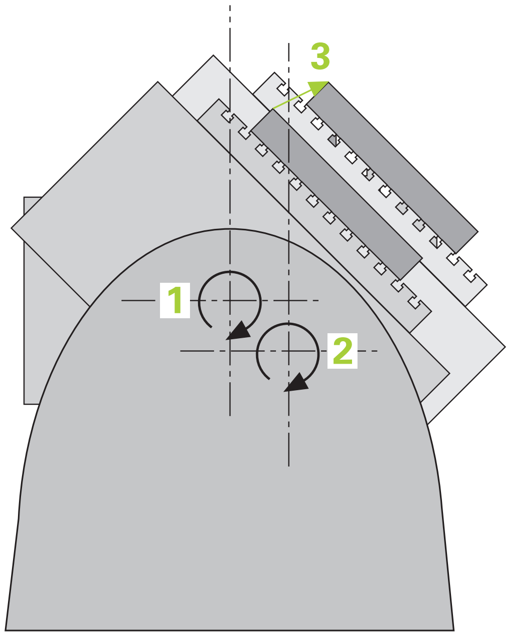
Touch Probe Cycles: Automatic Kinematics Measurement

20.1 Kinematics measurement with TS touch probes (KinematicsOpt option)

Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts need to be manufactured with precision and reproducible accuracy even over extended periods.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure at right) and the kinematic conditions actually existing on the machine (see **2** in the figure at right). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure at right). It is therefore necessary for the model to approach reality as closely as possible.

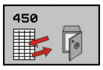


The **KinematicsOpt** function of the control is an important component that helps you to really meet these complex requirements: a 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, a calibration sphere is attached at any position on the machine table, and measured with a resolution that you define. During cycle definition, you simply define for each rotary axis the area that you want to measure.

From the measured values, the control calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics table.

Overview

The control offers the following cycles that enable you to automatically save, restore, check, and optimize the machine kinematics:

Soft key	Cycle	Page
	450 **SAVE KINEMATICS** Automatic saving and restoring of kinematic configurations	748
	451 **MEASURE KINEMATICS** Automatic checking or optimizing of machine kinematics	751
	452 **PRESET COMPENSATION** Automatic checking or optimizing of machine kinematics	766
	453 **KINEMATICS GRID** Automatic checking or optimizing of machine kinematics	776

20.2 Prerequisites

The following are prerequisites for using the KinematicsOpt option:

- The software options 48 (KinematicsOpt), 8 (Software option 1) and 17 (Touch Probe function) must be enabled.
- The 3-D touch probe used for the measurement must be calibrated.
- The cycles can only be carried out with the tool axis Z.
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table. HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.
- The kinematics description of the machine must be complete and correct. The transformation values must be entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine tool builder during commissioning).
- The machine tool builder must have defined the machine parameters for **CfgKinematicsOpt** (no. 204800) in the configuration data. **maxModification** (no. 204801) specifies the tolerance limit starting from which the control is to display a message if the changes made to the kinematic data exceed this limit value. **maxDevCalBall** (no. 204802) defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter. **mStrobeRotAxPos** (no. 204803) defines an M function that is specifically configured by the machine manufacturer and is used to position the rotary axes.

Please note while programming:

NOTICE

Danger of collision!

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

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

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

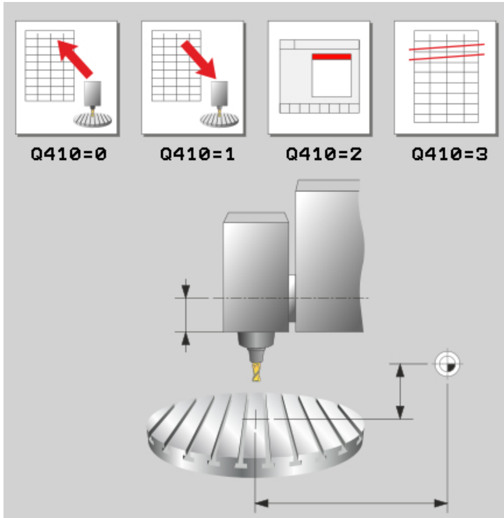
If an M function has been defined in machine parameter **mStrobeRotAxPos** (no. 204803), you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for 450).

If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

20.3 SAVE KINEMATICS (Cycle 450, ISO: G450, option)

Cycle run

With Touch Probe Cycle 450 you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.



Please note while programming:

Always save the active kinematics configuration before running a kinematics optimization. Advantage:

- You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).

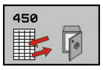
With the **Restore** mode, note the following:

- The control can restore saved data only to a matching kinematic configuration.
- A change in the kinematics always changes the preset as well. Define the preset again, if necessary.

The cycle will not restore identical values, but only data that is different from the existing data. Compensations can only be restored if they had been saved before.

Only save and restore data with Cycle 450, while no tool carrier kinematics configuration that includes transformations is active.

Cycle parameters



- **Q410 Mode (0/1/2/3)?**: Define whether you wish to backup or restore the kinematics configuration:
  0: Backup active kinematics configuration
  1: Restore saved kinematics configuration
  2: Display current memory status
  3: Delete a data record.
- **Q409/QS409 Name of data record?**: Number or name of the record designator. When specifying a number, you can enter values from 0 to 99999, and when entering letters the string length may not exceed 16 characters. 16 memory locations in total are available. Q409 has no function if mode 2 has been selected. Wildcards can be used for searches in modes 1 and 3 (Restore and Delete). If the control finds several possible data records because of the wildcards, the control restores the mean values of the data (mode 1) or deletes all data blocks after confirmation (mode 3). You can use the following wildcards for searching:
  ?: Any single character
  $: Any single alphabetic character (letter)
  #: Any single number
  *****: Any character string of any length

Saving the current kinematics

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=0     ;MODE
   Q409=947   ;MEMORY DESIGNATION
```

Restoring data blocks

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=1     ;MODE
   Q409=948   ;MEMORY DESIGNATION
```

Displaying all saved data blocks

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=2     ;MODE
   Q409=949   ;MEMORY DESIGNATION
```

Deleting data blocks

```
5 TCH PROBE 450  SAVE KINEMATICS
   Q410=3     ;MODE
   Q409=950   ;MEMORY DESIGNATION
```

Logging function

After running Cycle 450, the control creates a log (**tchpr450.txt**) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

The other data in the log vary depending on the selected mode:

- Mode 0: Logging of all axis entries and transformation entries of the kinematics chain that the control has saved.
- Mode 1: Logging of all transformation entries before and after restoring the kinematics configuration.
- Mode 2: List of the saved data records.
- Mode 3: List of the deleted data records.

Notes on data management

The control stores the saved data in the file **TNC:\table\DATA450.KD**. This file can be backed up to an external PC with **TNCremo**, for example. If you delete the file, the stored data is removed, too. If the data in the file is changed manually, the data records may become corrupted so that they are unusable.

> If the **TNC:\table\DATA450.KD** file does not exist, it is generated automatically when Cycle 450 is executed.
>
> Make sure that you delete empty files with the name **TNC:\table\DATA450.KD**, if any, before starting Cycle 450. If there is an empty storage table (**TNC:\table\DATA450.KD**) that does not contain any lines yet, an error message will be displayed upon starting to run Cycle 450. In this case, delete the empty storage table and call the cycle again.
>
> Do not change stored data manually.
>
> Make a backup of the **TNC:\table\DATA450.KD** file so that you can restore the file, if necessary (e.g. if the data medium is damaged).

20.4 MEASURE KINEMATICS (Cycle 451, ISO: G451, option)

Cycle run

Touch probe cycle 451 enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

> HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The control will determine the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

1 Clamp the calibration sphere and check for potential collisions.
2 In Manual mode of operation, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select the Program Run operating mode and start the calibration program.
4 The control automatically measures all rotary axes successively in the resolution you defined.
5 The control saves the measured values in the following Q parameters:

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (–1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (–1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. A reference measurement is automatically performed at 0°.

Specify the start and end angles in such a way that the same position is not measured twice. A duplicated point measurement (e.g. measuring positions +90° and -270°) is not advisable, but it will not generate an error message.

- Example: Start angle = +90°, end angle = –90°
  - Start angle = +90°
  - End angle = –90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (-90° - +90°) / (4 – 1) = -60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +30°
  - Measuring point 3 = -30°
  - Measuring point 4 = -90°
- Example: start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (270° – 90°) / (4 – 1) = +60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +150°
  - Measuring point 3 = +210°
  - Measuring point 4 = +270°

Machines with Hirth-coupled axes

NOTICE

Danger of collision!

In order to be positioned, the axis must move out of the Hirth grid. If necessary, the control rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points).

- So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration sphere
- Also ensure that there is enough space to reach the safety clearance (software limit switch)

NOTICE

Danger of collision!

Depending on the machine configuration, the control cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine tool builder, enabling the control to move the rotary axes. The machine tool builder must have entered the number of the M function in machine parameter **mStrobeRotAxPos** (no. 244803) for this purpose.

- Note the documentation of the machine tool builder

Define a retraction height greater than 0 if software option 2 is not available.

The measured positions are calculated from the start angle, end angle and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = –30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = (Q412 – Q411) / (Q414 – 1)

Calculated stepping angle = (90° - (-30°)) / (4 – 1) = 120 / 3 = 40°

Measuring position 1 = Q411 + 0 * stepping angle = -30° –> -30°

Measuring position 2 = Q411 + 1 * stepping angle = +10° –> 9°

Measuring position 3 = Q411 + 2 * stepping angle = +50° –> 51°

Measuring position 4 = Q411 + 3 * stepping angle = +90° –> 90°

Defining the number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1- 2), e.g. when commissioning the machine.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°. Thus, define a starting angle of 90° and an end angle of 270°.

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the **Check** mode.

> If a measuring point has been defined at 0°, it will be ignored because the reference measurement is always done at 0°.

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machine with rotary tables/tilting tables: Clamp the calibrating ball as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

Notes on the accuracy

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion of results recorded by the control in the measuring log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.

> If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.
>
> If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine tool manual provides further information.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
  - Position the calibration sphere in such a way on the machine table that with rotary table axes, there is a large measuring circle or that on swivel head axes, measurement can be made at a representative position (e.g. in the center of the traverse range).
- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made near the rotary axis angle at which the workpiece is to be machined.
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.
- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes.

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is made with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432**, you can activate backlash measurement. Enter an angle that the control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.

> The control does not perform an automatic backlash compensation.
>
> If the measuring circle radius is < 1 mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can determine the rotary axis backlash (see "Logging function", Page 765).
>
> Backlash measurement is not possible if an M function for positioning the rotary axes is set in machine parameter **mStrobeRotAxPos** (no. 204803) or if the axis is a Hirth axis.

Please note while programming:

Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.

As with Cycles 451 and 452, Cycle 453 ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or set input parameter Q431 to 1 or 3, respectively.

If machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to -1 (M function positions the rotary axes), only start a measurement if all rotary axes are at 0°.

For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.

The control ignores cycle definition data that applies to inactive axes.

For angle optimization, the machine manufacturer must change the configuration correspondingly.

A correction in the machine datum (Q406=3) is only possible if superimposed rotary axes on the spindle head side or table side are measured.

Compensation of the angle is only possible with option #52 **KinematicsComp**.

If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification**), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Programming in inches: The control always records the log data and results of measurement in millimeters.

During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

Cycle parameters

- **Q406 Mode (0/1/2/3)?**: Specify whether the control should check or optimize the active kinematics:
  0: Check active machine kinematics. The control measures the kinematics in the rotary axes you have defined, but it does not make any changes to it. The control displays the results of measurement in a measurement log.
  1: Optimize active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **the position of the rotary axes** of the active kinematics.
  2: Optimize active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **angle and positioning errors.** Software option 52, KinematicsComp, is required for the compensation of angle errors.
  3: Optimize active machine kinematics: The control automatically corrects the machine datum. It then optimizes **angle and positioning errors.** Software option 52, KinematicsComp, is required.
- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999Alternatively **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. Less measuring points increase speed and more measuring points increase measurement precision.
- **Q431 Preset (0/1/2/3)?** Define whether the control automatically sets the active preset to the center of the sphere:
  0: Do not define the preset automatically at the center of the sphere: Preset manually before cycle start
  1: Define the preset automatically at the center of the sphere (this overwrites the active preset): Manually preposition the touch probe before cycle start using the calibration sphere
  2: Automatically define the preset at the center of the sphere after measuring): Preset manually before cycle start
  3: Define the preset before and after measurement at the center of the sphere (the active preset will be overwritten): Preposition the touch probe manually before cycle start using the calibration sphere
- **Q432 Angular range of backlash comp.?**: Here, you define the angle value to be used as traversing angle when measuring the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input range: -3.0000 to +3.0000

> If you have activated presetting before the calibration (Q431 = 1/3), then move the touch probe to the set-up clearance (Q320 + SET_UP) to a position approximately above the center of the calibration sphere before the start of the cycle.

Various modes (Q406)

Test mode Q406 = 0

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control records the results of a possible position optimization but does not make any adjustments.

"Optimize position of rotary axes" mode Q406 = 1

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the control tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data is adjusted automatically.

Position and Angle Optimization mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- First the control tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

```
1 TOOL CALL  "TCH PROBE" Z
2 TCH PROBE 451  MEASURE
   KINEMATICS
   Q406=1      ;MODE
   Q407=12.5   ;SPHERE RADIUS
   Q320=0      ;SET-UP CLEARANCE
   Q408=0      ;RETR. HEIGHT
   Q253=750    ;F PRE-POSITIONING
   Q380=0      ;REFERENCE ANGLE
   Q411=-90    ;START ANGLE A AXIS
   Q412=+90    ;END ANGLE A AXIS
   Q413=0      ;INCID. ANGLE A AXIS
   Q414=0      ;MEAS. POINTS A AXIS
   Q415=-90    ;START ANGLE B AXIS
   Q416=+90    ;END ANGLE B AXIS
   Q417=0      ;INCID. ANGLE B AXIS
   Q418=0      ;MEAS. POINTS B AXIS
   Q419=+90    ;START ANGLE C AXIS
   Q420=+270   ;END ANGLE C AXIS
   Q421=0      ;INCID. ANGLE C AXIS
   Q422=3      ;MEAS. POINTS C AXIS
   Q423=3      ;NO. OF PROBE POINTS
   Q431=1      ;PRESET
   Q432=0.5    ;BACKLASH, ANG. RANGE
```

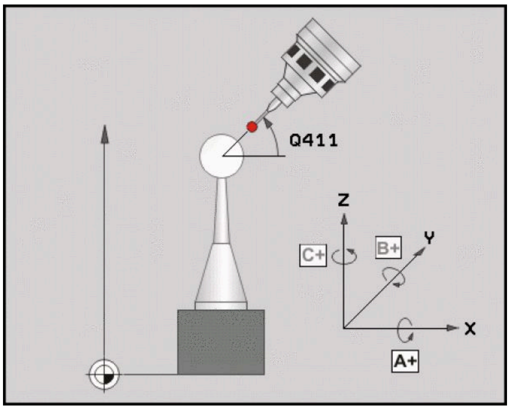
Logging function

After running Cycle 451, the control will create a log **(TCHPR451.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

20.5 PRESET COMPENSATION (Cycle 452, ISO: G452, option)

Cycle run



Touch probe cycle 452 optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, ISO: G451, option)", Page 751). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the calibration sphere after optimization.

This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

1 Clamp the calibration sphere
2 Measure the complete reference head with Cycle 451, and use Cycle 451 to finally set the preset in the center of the sphere.
3 Insert the second head.
4 Use Cycle 452 to measure the interchangeable head up to the point where the head is changed.
5 Use Cycle 452 to adjust other interchangeable heads to the reference head.

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

1 Clamp the calibration sphere and check for potential collisions.
2 Set the preset in the calibration sphere.
3 Set the preset on the workpiece, and start machining the workpiece.
4 Use Cycle 452 for preset compensation at regular intervals. The control measures the drift of the axes involved and compensates it in the kinematics description.

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (–1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (–1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Please note while programming:

Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.

As with Cycles 451 and 452, Cycle 453 ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.

In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

Note that all functions for tilting in the working plane are reset.

Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle, you must set the preset in the center of the calibration sphere and activate it.

For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse an angle of 1° to the limit switch. The control needs this traverse for internal backlash compensation.

For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

If the kinematics data determined exceed the permissible limit (**maxModification**), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Programming in inches: The control always records the log data and results of measurement in millimeters.

Cycle parameters

- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999
- **Q408 Retraction height?** (absolute): Input range: 0.0001 to 99999.9999
  0: Do not move to retraction height. The control moves the touch probe to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C
  >0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253.
- **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min. Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q380 Ref. angle in ref. axis?** (absolute) Enter the Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. Input range: 0 to 360.0000
- **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made. Input range: -359.999 to 359.999
- **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made. Input range: -359.999 to 359.999
- **Q413 Angle of incidence in A axis?**: Angle of incidence in the A axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999
- **Q414 No. of meas. points in A (0...12)?**: Number of probe measurements to be used for measuring the A axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12

Calibration program

4 TOOL CALL "TCH PROBE" Z	
5 TCH PROBE 450 SAVE KINEMATICS	
Q410=0	;MODE
Q409=5	;MEMORY DESIGNATION
6 TCH PROBE 452 PRESET COMPENSATION	
Q407=12.5	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q408=0	;RETR. HEIGHT
Q253=750	;F PRE-POSITIONING
Q380=0	;REFERENCE ANGLE
Q411=-90	;START ANGLE A AXIS
Q412=+90	;END ANGLE A AXIS
Q413=0	;INCID. ANGLE A AXIS
Q414=0	;MEAS. POINTS A AXIS
Q415=-90	;START ANGLE B AXIS
Q416=+90	;END ANGLE B AXIS
Q417=0	;INCID. ANGLE B AXIS
Q418=2	;MEAS. POINTS B AXIS
Q419=-90	;START ANGLE C AXIS
Q420=+90	;END ANGLE C AXIS
Q421=0	;INCID. ANGLE C AXIS
Q422=2	;MEAS. POINTS C AXIS
Q423=4	;NO. OF PROBE POINTS
Q432=0	;BACKLASH, ANG. RANGE

- **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made. Input range: -359.999 to 359.999
- **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made. Input range: -359.999 to 359.999
- **Q417 Angle of incidence in B axis?**: Inclination angle in the B axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999
- **Q418 No. of meas. points in B (0...12)?**: Number of probe measurements to be used for measuring the B axis. If the input value = 0, the control does not measure the respective axis. Input range: 0 to 12
- **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made. Input range: -359.999 to 359.999
- **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made. Input range: -359.999 to 359.999
- **Q421 Angle of incidence in C axis?**: Inclination angle in the C axis at which the other rotary axes are to be measured. Input range: -359.999 to 359.999
- **Q422 No. of meas. points in C (0...12)?**: Number of probe measurements to be used for measuring the C axis. Input range: 0 to 12. If the input value = 0, the control does not measure the respective axis.
- **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Input range: 3 to 8. Less measuring points increase speed and more measuring points increase measurement precision.
- **Q432 Angular range of backlash comp.?**: Here, you define the angle value to be used as traversing angle when measuring the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input range: -3.0000 to +3.0000

Adjustment of interchangeable heads

The goal of this procedure is to achieve that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, a fork head is adjusted to the A and C axes. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- Insert the interchangeable head that will be used as a reference head.
- Clamp the calibration sphere
- Insert the touch probe
- Use Cycle 451 to measure the complete kinematics, including the reference head.
- Set preset (using Q431 = 2 or 3 in Cycle 451) after measuring the reference head.

Measuring a reference head

```
1 TOOL CALL "TCH PROBE" Z
2 TCH PROBE 451 MEASURE
    KINEMATICS
    Q406=1      ;MODE
    Q407=12.5   ;SPHERE RADIUS
    Q320=0      ;SET-UP CLEARANCE
    Q408=0      ;RETR. HEIGHT
    Q253=2000   ;F PRE-POSITIONING
    Q380=+45    ;REFERENCE ANGLE
    Q411=-90    ;START ANGLE A AXIS
    Q412=+90    ;END ANGLE A AXIS
    Q413=45     ;INCID. ANGLE A AXIS
    Q414=4      ;MEAS. POINTS A AXIS
    Q415=-90    ;START ANGLE B AXIS
    Q416=+90    ;END ANGLE B AXIS
    Q417=0      ;INCID. ANGLE B AXIS
    Q418=2      ;MEAS. POINTS B AXIS
    Q419=+90    ;START ANGLE C AXIS
    Q420=+270   ;END ANGLE C AXIS
    Q421=0      ;INCID. ANGLE C AXIS
    Q422=3      ;MEAS. POINTS C AXIS
    Q423=4      ;NO. OF PROBE POINTS
    Q431=3      ;PRESET
    Q432=0      ;BACKLASH, ANG. RANGE
```

- Load the second interchangeable head.
- Insert the touch probe
- Measure the interchangeable head with Cycle 452
- Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with Q422)
- The preset and the position of the calibration sphere must not be changed during the entire process.
- All other interchangeable heads can be adjusted in the same way

The head change function can vary depending on the individual machine tool. Refer to your machine manual.

Adjusting an interchangeable head

```
3 TOOL CALL "TCH PROBE" Z
4 TCH PROBE 452 PRESET
  COMPENSATION
  Q407=12.5  ;SPHERE RADIUS
  Q320=0     ;SET-UP CLEARANCE
  Q408=0     ;RETR. HEIGHT
  Q253=2000  ;F PRE-POSITIONING
  Q380=+45   ;REFERENCE ANGLE
  Q411=-90   ;START ANGLE A AXIS
  Q412=+90   ;END ANGLE A AXIS
  Q413=45    ;INCID. ANGLE A AXIS
  Q414=4     ;MEAS. POINTS A AXIS
  Q415=-90   ;START ANGLE B AXIS
  Q416=+90   ;END ANGLE B AXIS
  Q417=0     ;INCID. ANGLE B AXIS
  Q418=2     ;MEAS. POINTS B AXIS
  Q419=+90   ;START ANGLE C AXIS
  Q420=+270  ;END ANGLE C AXIS
  Q421=0     ;INCID. ANGLE C AXIS
  Q422=0     ;MEAS. POINTS C AXIS
  Q423=4     ;NO. OF PROBE POINTS
  Q432=0     ;BACKLASH, ANG. RANGE
```

Drift compensation

During machining various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle 452.

- Clamp the calibration sphere
- Insert the touch probe
- Measure the complete kinematics with Cycle 451 before starting the machining process
- Set the preset (using Q432 = 2 or 3 in Cycle 451) after measuring the kinematics.
- Then set the presets on your workpiece and start the machining process.

Reference measurement for drift compensation

```
1 TOOL CALL "TCH PROBE" Z
2 CYCL DEF 247 PRESETTING
  Q339=1      ;PRESET NUMBER
3 TCH PROBE 451 MEASURE
  KINEMATICS
  Q406=1      ;MODE
  Q407=12.5   ;SPHERE RADIUS
  Q320=0      ;SET-UP CLEARANCE
  Q408=0      ;RETR. HEIGHT
  Q253=750    ;F PRE-POSITIONING
  Q380=+45    ;REFERENCE ANGLE
  Q411=+90    ;START ANGLE A AXIS
  Q412=+270   ;END ANGLE A AXIS
  Q413=45     ;INCID. ANGLE A AXIS
  Q414=4      ;MEAS. POINTS A AXIS
  Q415=-90    ;START ANGLE B AXIS
  Q416=+90    ;END ANGLE B AXIS
  Q417=0      ;INCID. ANGLE B AXIS
  Q418=2      ;MEAS. POINTS B AXIS
  Q419=+90    ;START ANGLE C AXIS
  Q420=+270   ;END ANGLE C AXIS
  Q421=0      ;INCID. ANGLE C AXIS
  Q422=3      ;MEAS. POINTS C AXIS
  Q423=4      ;NO. OF PROBE POINTS
  Q431=3      ;PRESET
  Q432=0      ;BACKLASH, ANG. RANGE
```

- Measure the drift of the axes at regular intervals.
- Insert the touch probe
- Activate the preset in the calibration sphere.
- Use Cycle 452 to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the entire process.

> This procedure can also be performed on machines without rotary axes.

Drift compensation

```
4 TOOL CALL "TCH PROBE" Z
5 TCH PROBE 452 PRESET
  COMPENSATION
  Q407=12.5   ;SPHERE RADIUS
  Q320=0      ;SET-UP CLEARANCE
  Q408=0      ;RETR. HEIGHT
  Q253=99999;F PRE-POSITIONING
  Q380=+45    ;REFERENCE ANGLE
  Q411=-90    ;START ANGLE A AXIS
  Q412=+90    ;END ANGLE A AXIS
  Q413=45     ;INCID. ANGLE A AXIS
  Q414=4      ;MEAS. POINTS A AXIS
  Q415=-90    ;START ANGLE B AXIS
  Q416=+90    ;END ANGLE B AXIS
  Q417=0      ;INCID. ANGLE B AXIS
  Q418=2      ;MEAS. POINTS B AXIS
  Q419=+90    ;START ANGLE C AXIS
  Q420=+270   ;END ANGLE C AXIS
  Q421=0      ;INCID. ANGLE C AXIS
  Q422=3      ;MEAS. POINTS C AXIS
  Q423=3      ;NO. OF PROBE POINTS
  Q432=0      ;BACKLASH, ANG. RANGE
```

Logging function

After running Cycle 452, the control creates a log (**TCHPR452.html**) containing the following information:

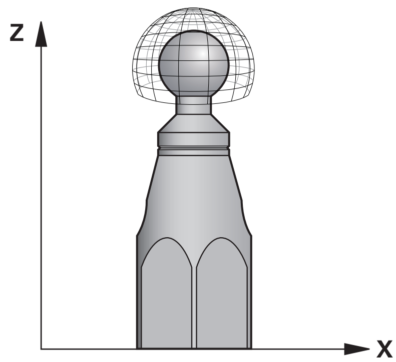
- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes
  - Position before preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

Notes on log data

(see "Logging function", Page 765)

20.6 KINEMATICS GRID (Cycle 453, ISO: G453, option)

Cycle run



Even if your machine was already optimized regarding positioning errors (e.g. via Cycle 451), residual errors at the Tool Center Point (TCP) during tilting of the rotary axes may remain. Such errors occur particularly with swivel-head machines. These can result for example from component errors (e.g. a bearing error) with head rotary axes.

Cycle 453 KINEMATICS GRID enables these errors to be determined and compensated in accordance with the tilting axis positions Software options 48, KinematicsOpt, and 52, KinematicsComp, are required. With this cycle and using a 3-D TS touch probe, you measure a HEIDENHAIN calibration sphere that you have attached to the machine table. The cycle then moves the touch probe automatically to positions in a grid-line arrangement around the calibration sphere. The machine tool builder defines these tilting axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing process on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine tool builder defines this compensation table (*.kco) and specifies its storage location.

If you work with Cycle 453 you execute the cycle at several different positions in the working space. You can check immediately whether the compensation with Cycle 453 has the desired positive effects on machine accuracy. This type of compensation for the specific machine is only suitable if the desired improvements are achieved with identical compensation values at several positions. If this is not the case, the errors should be searched for outside of the rotary axes.

Perform the measurement with Cycle 453 with optimized rotary axis positioning error. For this purpose previously work with e.g. Cycle 451.

> 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.
>
> For Cycle 453 KINEMATICS GRID the software options 48, KinematicsOpt, and 52, KinematicsComp, are required.

> This function must be enabled and adapted by the machine tool builder.
>
> To use this cycle, your machine tool builder needs to create and configure a compensation table (*kco) first and perform some more settings.

The control then optimizes the accuracy of your machine. For this purpose, it automatically saves the compensation values resulting from a measurement in a compensation table (*kco). (This applies to mode Q406=1.)

1 Clamp the calibration sphere and check for potential collisions.
2 In Manual mode of operation, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select Program Run mode and start the NC program.
4 The cycle is run in accordance with Q406 (-1 = delete / 0 = check / 1 = compensate)

Various modes (Q406)

Delete mode Q406 = -1

- The axes are not moved
- The TNC writes all values to the compensation table (*kco), setting them to "0". The result is that no further compensations will be effective for the currently selected Kinematics.

Test mode Q406 = 0

- The control probes the calibration sphere.
- The results are saved to a log in .html format. This log is stored in the directory where the current NC program resides.

Compensate mode Q406 = 1

- The control probes the calibration sphere.
- The control writes the deviations to the compensation table (*kco). The table is updated, the compensations are immediately effective.
- The results are saved to a log in .html format. This log is stored in the directory where the current NC program resides.

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. It is recommended to clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

Please note while programming:

For Cycle 453 KINEMATICS GRID the software options 48, KinematicsOpt, and 52, KinematicsComp, are required.

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

The machine tool builder defines the storage location of the compensation table (*kco).

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 you set input parameter Q431 to 1 or 3, respectively.

If machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to -1 (M function positions the rotary axes), only start a measurement if all rotary axes are at 0°.

For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.

When probing, the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in machine parameter **maxDevCalBall** (no. 204802), the control repeats the measurement and only then displays an error message and ends the measurement.

Programming in inches: The control always records the log data and results of measurement in millimeters.

During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

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.

Cycle parameters

- **Q406 Mode (-1/0/+1)** : Define whether the control should write the values to the compensation table (*.kco) with 0, check the currently existing deviations, or perform a compensation. A log (*.html) is created.
  -1: Delete values in the compensation table (*.kco). The compensation values for TCP positioning errors are set to 0 in the compensation table (*.kco). The control will not perform any probing. No results will be output to the log (*.html).
  0: Check TCP positioning errors. The control measures the TCP positioning errors in accordance with the rotary axis positions but does not write values to the compensation table (*.kco). The control displays the standard and maximum deviation in a log (*.html).
  1: Compensate TCP positioning errors. The control measures the TCP positioning errors in accordance with the rotary axis positions and writes the deviations to the compensation table (*.kco). The compensations are then immediately effective. The control displays the standard and maximum deviation in a log (*.html).
- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used. Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** Define an additional distance between measuring point and ball tip. Q320 is added to **SET_UP** in the touch probe table. Input range: 0 to 99999.9999Alternatively **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.

Probing with Cycle 453

```
4 TOOL CALL "TCH PROBE" Z
6 TCH PROBE 453 KINEMATICS GRID
    Q406=0      ;MODE
    Q407=12.5   ;SPHERE RADIUS
    Q320=0      ;SET-UP CLEARANCE
    Q408=0      ;RETR. HEIGHT
    Q253=750    ;F PRE-POSITIONING
    Q380=0      ;REFERENCE ANGLE
    Q423=4      ;NO. OF PROBE POINTS
    Q431=0      ;PRESET
```

- **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
- **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Input range: 3 to 8. Less measuring points increase speed and more measuring points increase measurement precision.
- **Q431 Preset (0/1/2/3)?** Define whether the control automatically sets the active preset to the center of the sphere:
  0: Do not define the preset automatically at the center of the sphere: Preset manually before cycle start
  1: Define the preset automatically at the center of the sphere (this overwrites the active preset): Manually preposition the touch probe before cycle start using the calibration sphere
  2: Automatically define the preset at the center of the sphere after measuring): Preset manually before cycle start
  3: Define the preset before and after measurement at the center of the sphere (the active preset will be overwritten): Preposition the touch probe manually before cycle start using the calibration sphere

> 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.

Logging function

After running Cycle 453, the control will create a log **(TCHPR453.html**) and save it in the folder where the current NC program resides. It contains the following data:

- Date and time of protocol creation
- Path of the NC program from which the cycle was run
- Number and name of the currently active tool
- Mode
- Measured data: Standard deviation and maximum deviation
- Information at which position in degrees (°) the maximum deviation occurred
- Number of measuring positions

21

Touch Probe Cycles: Automatic Tool Measurement

21.1 Fundamentals

Overview

Operating notes

- When running touch probe cycles, Cycle **8 MIRROR IMAGE**, Cycle **11 SCALING**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active.
- HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.

The control and the machine tool must be set up by the machine tool builder for use of the TT touch probe.

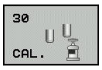
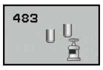
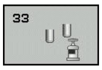
Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.

The touch probe cycles are available only with the Touch Probe Functions software option (option 17).

In conjunction with the control's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central TOOL.T tool file and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Measurement of a stationary tool
- Measurement of a rotating tool
- Measurement of individual teeth

You can program the cycles for tool measurement in **Programming** mode of operation using the **TOUCH PROBE** key. The following cycles are available:

New format	Old format	Cycle	Page
		Calibrating the TT, Cycles 30 and 480	790
		Calibrating the wireless TT 449, Cycle 484	792
		Measuring the tool length, Cycles 31 and 481	794
		Measuring the tool radius, Cycles 32 and 482	796
		Measuring the tool length and radius, Cycles 33 and 483	798

> 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

> Before you start working with the measuring cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200).
>
> Touch probe cycles 480, 481, 482, 483 and 484 can be hidden with the machine parameter **hideMeasureTT** (No. 128901).
>
> When measuring a stationary tool, the Control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius [mm]

The probing feed rate is calculated as follows:
v = measuring tolerance • n with

v:	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate:

probingFeedCalc (No. 122710) = **ConstantTolerance**:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance (**measureTolerance1** (no. 122715), the sooner you will encounter this effect.

probingFeedCalc (No. 122710) = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc (No. 122710) = **ConstantFeed**:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance = r • **measureTolerance1**/ 5 mm, where

r:	Active tool radius [mm]
measureTolerance1:	Maximum permissible error of measurement

Entries in the tool table TOOL.T

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT.	Cutting direction of the tool for measuring a rotating tool	**Cutting direction (M3 = -)?**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to **offsetToolAxis**. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill	4 (4 teeth)	R (offset required because tool diameter is larger than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from **offsetToolAxis** (no. 122707) used)
Radius cutter e.g. with a diameter of 10 mm	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured	5 (always define an offset at least equal to the tool radius in order to make sure that the measured diameter is correct)

21.2 Calibrating the TT (Cycle 30 or 480, ISO: G480 option 17)

Cycle run

The TT is calibrated with measuring cycle TCH PROBE 30 or TCH PROBE 480. (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 785). The calibration process runs automatically. The control also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylindrical pin. The resulting calibration values are stored in the control memory and are accounted for during subsequent tool measurement.

Calibration process:

1. Clamp the calibrating tool. The calibrating tool must be a precisely cylindrical part, for example a cylinder pin
2. Manually position the calibrating tool in the working plane via the center of the TT
3. Position the calibrating tool in the tool axis approx. 15 mm + safety clearance above the TT
4. The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
5. The calibration process along the tool axis starts
6. Calibration then follows in the working plane
7. The control positions the calibrating tool in the working plane at a position of TT radius + set-up clearance + 11 mm
8. Then the TNC moves the tool downwards along the tool axis and the calibration process starts
9. During probing, the control moves in a square pattern
10. The control saves the calibration values and considers them during subsequent tool measurement
11. The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Please note while programming:

The functioning of the calibration cycle is dependent on machine parameter **CfgTTRoundStylus** (No. 114200). Refer to your machine manual.

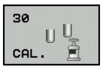
The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the TOOL.T tool table.

The position of the TT within the machine working space must be defined by setting machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**.

If you change the setting of any of the machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**, you must recalibrate.

Cycle parameters



- **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistToolAx** (no. 114203)). Input range: -99999.9999 to 99999.9999

Example of 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
```

21.3 Calibrating the wireless TT 449 (Cycle 484, ISO: G484)

Fundamentals

With Cycle 484, you can calibrate your tool touch probe, e.g the wireless infrared TT 449 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle 484 you must move the tool over the TT

Cycle run

To calibrate the tool touch probe, program measuring cycle TCH PROBE 484. In input parameter Q536, you can specify whether you want to run the cycle semi-automatically or fully automatically.

Semi-automatic—stop before running

- Insert the calibrating tool
- Define and start the calibration cycle
- The control interrupts the calibration cycle.
- The control displays a dialog in a new window.
- You are prompted to manually position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact.

Fully automatic—no stop before running

- Insert the calibrating tool
- Position the calibrating tool above the center of the touch probe. Ensure that the calibrating tool is located above the measuring surface of the probe contact.
- Define and start the calibration cycle
- The calibration cycle is executed without stopping. The calibration process starts from the current position of the tool.

Calibrating tool:

The calibrating tool must be a precisely cylindrical part, for example a cylindrical pin. Enter the exact length and radius of the calibrating tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Please note while programming:

NOTICE

Danger of collision!

To avoid collisions the tool must be pre-positioned before calling the cycle with **Q536**=1! The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

- Specify whether to stop before cycle start or run the cycle automatically without stopping.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

The calibrating tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. When using a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. The use of a calibrating tool of too small a diameter and/or protruding too far from the chuck may cause significant inaccuracies.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The TT needs to be recalibrated if you change its position on the table.

Cycle parameters

- **Q536 Stop before running (0=Stop)?**: Specify whether to stop before cycle start or run the cycle automatically without stopping:
  0: Stop before running the cycle. You are prompted in a dialog to manually position the tool above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC Start to continue the calibration process or press the **CANCEL** soft key to cancel the calibration process
  1: No stop before running the cycle. The control starts the calibration process from the current position. Before running Cycle 484, you must position the tool above the tool touch probe.

Example

```
6 TOOL CALL 1 Z
7 TCH PROBE 484 CALIBRATE TT
    Q536=+0     ;STOP BEFORE RUNNING
```

21.4 Measuring tool length (Cycle 31 or 481, ISO: G481)

Cycle run

To measure the tool length, program the measuring cycle TCH PROBE 31 or TCH PROBE 481 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483"). Via input parameters you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is stationary.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is stationary.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**R-OFFS**).

Cycle for measuring a stationary tool (e.g. for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

Cycle for measuring individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). You can enter an additional offset in Tool offset: Length (**L-OFFS**) in the tool table. The control probes the tool radially while it is rotating to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function in Cycle TCH PROBE 31, set parameter Probe the teeth = 1.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

You can run an individual tooth measurement of tools with **up to 20 teeth**.

Cycle parameters

- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T)
  2: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q115. Nothing is entered under L or DL in the tool table.
- **Parameter number for result?**: Parameter number in which the control saves the status of measurement results:
  0.0: Tool is within tolerance
  1.0: Tool is worn (**LTOL** exceeded)
  2.0: Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement in your NC program, answer the dialog prompt with **NO ENT**.
- **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: -99999.9999 to 99999.9999
- **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 31.0  CAL. TOOL LENGTH
8 TCH PROBE 31.1  CHECK: 0
9 TCH PROBE 31.2  HEIGHT: +120
10 TCH PROBE 31.3  PROBING THE
   TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL  12 Z
7 TCH PROBE 31.0  CAL. TOOL LENGTH
8 TCH PROBE 31.1  CHECK: 1  q5
9 TCH PROBE 31.2  HEIGHT: +120
10 TCH PROBE 31.3  PROBING THE
   TEETH: 1
```

Example of new format

```
6 TOOL CALL  12 Z
7 TCH PROBE 481  CAL. TOOL LENGTH
   Q340=1       ;CHECK
   Q260=+100 ;CLEARANCE HEIGHT
   Q341=1       ;PROBING THE TEETH
```

21.5 Measuring a tool radius (Cycle 32 or 482, ISO: G482)

Cycle run

To measure a tool radius, program the measuring cycle TCH PROBE 32 or TCH PROBE 482 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 785). Select via input parameters by which of two methods the tool radius is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. The control probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Please note while programming:

Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, define the number of teeth **CUT** as 0 in the tool table and adjust machine **CfgTT** (no. 122700). Refer to your machine manual.

Cycle parameters

- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T)
  2: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q116. Nothing is entered under R or DR in the tool table.
- **Parameter number for result?**: Parameter number in which the control saves the status of the measurement results:
  0.0: Tool is within tolerance
  1.0: Tool is worn (**RTOL** exceeded)
  2.0: Tool is broken (**RBREAK** exceeded). If you do not wish to use the result of measurement in your NC program, answer the dialog prompt with **NO ENT**.
- **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: -99999.9999 to 99999.9999
- **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).

Measuring a rotating tool for the first time; old format

6 TOOL CALL 12 Z
7 TCH PROBE 32.0 CAL. TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 0

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

6 TOOL CALL 12 Z
7 TCH PROBE 32.0 CAL. TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 1

Example of new format

6 TOOL CALL 12 Z	
7 TCH PROBE 482 CAL. TOOL RADIUS	
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH

21.6 Measuring tool length and radius (Cycle 33 or 483, ISO: G483)

Cycle run

To measure both the length and radius of a tool, program the measuring cycle TCH PROBE 33 or TCH PROBE 483 (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 785). 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.

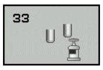
Please note while programming:

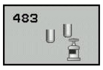
Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The functioning of the cycle is dependent on machine parameter **probingCapability** (No. 122723). (This parameter permits e.g. tool length measurement with a stationary spindle to be enabled and tool radius- and individual tooth measurement to be simultaneously disabled.) Refer to your machine manual.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgTT** (No. 122700) Refer to your machine manual.

Cycle parameters





- **Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.
  0: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL=0 and DR=0. If there is already a value in TOOL.T, it will be overwritten.
  1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in Q parameters Q115 and Q116. If the delta value is greater than the permissible tool length or radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T)
  2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The control calculates the deviation from the stored values and writes it to Q parameter Q115 or Q116. Nothing is entered under L, R, DL, or DR in the tool table.
- **Parameter number for result?**: Parameter number in which the control saves the status of the measurement results:
  0.0: Tool is within tolerance
  1.0: Tool is worn (**LTOL** and/or **RTOL** exceeded)
  2,0: Tool is broken (**LBREAK** and/or **RBREAK** exceeded). If you do not wish to use the result of measurement in your NC program, answer the dialog prompt with **NO ENT**.
- **Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input range: -99999.9999 to 99999.9999
- **Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).

Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 0
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE
   TEETH: 0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 1 q5
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE
   TEETH: 1
```

Example of new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 483 MEASURE TOOL
   Q340=1      ;CHECK
   Q260=+100   ;CLEARANCE HEIGHT
   Q341=1      ;PROBING THE TEETH
```

22

Tables of Cycles

22.1 Overview

Fixed cycles

Cycle number	Cycle name	DEF active	CALL active	Page
7	Datum shift	■		305
8	Mirroring	■		312
9	Dwell time	■		331
10	Rotation	■		314
11	Scaling factor	■		316
12	Program call	■		332
13	Oriented spindle stop	■		333
14	Contour definition	■		231
18	Thread cutting		■	371
19	Tilting the working plane	■		319
20	Contour data SL II	■		236
21	Pilot drilling SL II		■	238
22	Rough out SL II		■	240
23	Floor finishing SL II		■	244
24	Side finishing SL II		■	246
25	Contour train		■	249
26	Axis-specific scaling	■		317
27	Cylinder surface		■	273
28	Cylindrical surface slot		■	276
29	Cylinder surface ridge		■	280
32	Tolerance	■		334
39	Cylinder surface contour		■	283
200	Drilling		■	83
201	Reaming		■	85
202	Boring		■	87
203	Universal drilling		■	90
204	Back boring		■	96
205	Universal pecking		■	100
206	Tapping with a floating tap holder, new		■	125
207	Rigid tapping, new		■	128
208	Bore milling		■	108
209	Tapping with chip breaking		■	133
220	Polar pattern	■		219
221	Cartesian pattern	■		222

Cycle number	Cycle name	DEF active	CALL active	Page
225	Engraving		■	358
232	Face milling		■	364
233	Face milling (selectable milling direction, consider the side walls)		■	204
239	Ascertain the load	■		369
240	Centering		■	81
241	Single-lip deep-hole drilling		■	111
247	Presetting	■		311
251	Rectangular pocket (complete machining)		■	165
252	Circular pocket (complete machining)		■	171
253	Slot milling		■	178
254	Circular slot		■	183
256	Rectangular stud (complete machining)		■	189
257	Circular stud (complete machining)		■	194
258	Polygon stud		■	198
262	Thread milling		■	140
263	Thread milling/countersinking		■	144
264	Thread drilling/milling		■	148
265	Helical thread drilling/milling		■	152
267	Outside thread milling		■	156
270	Contour train data		■	258
275	Trochoidal slot		■	259
276	3-D contour train		■	253
285	Define gear	■		376
286	Gear hobbing		■	379
287	Gear skiving		■	385
291	Coupling turning interpolation		■	338
292	Contour turning interpolation		■	350

Turning cycles

Cycle number	Cycle name	DEF active	CALL active	Page
800	Adapt rotary coordinate system	■		408
801	Reset the turning system	■		415
810	Turn contour, longitudinal		■	429
811	Turn shoulder, longitudinal		■	418
812	Turn shoulder, longitudinal extended		■	420
813	Turn, longitudinal plunge		■	423
814	Turn, longitudinal plunge extended		■	426
815	Turn contour-parallel		■	433
820	Turn contour, transverse		■	449
821	Turn shoulder face		■	436
822	Turn shoulder face extended		■	438
823	Turn, transverse plunge		■	442
824	Turn, transverse plunge extended		■	445
830	Thread, contour-parallel		■	507
831	Thread, longitudinal		■	498
832	Thread, extended		■	502
840	Radial contour recessing		■	460
841	Simple radial recessing		■	453
842	Extended radial recessing		■	456
850	Axial contour recessing		■	471
851	Simple axial recessing		■	464
852	Extended axial recessing		■	467
860	Recessing contour, radial		■	482
861	Recessing, radial		■	475
862	Recessing, radial extended		■	478
870	Recessing contour, axial		■	494
871	Recessing, axial		■	486
872	Recessing, axial extended		■	489
880	Gear hobbing		■	516
883	Turning, simultaneous finishing		■	511
892	Check unbalance	■		523

Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	Reference plane	■		654
1	Polar preset	■		655
3	Measuring	■		695
4	Measuring in 3-D	■		697
444	Probing in 3D	■		699
30	Calibrate the TT	■		790
31	Measure/Inspect the tool length	■		794
32	Measure/Inspect the tool radius	■		796
33	Measure/Inspect the tool length and the tool radius	■		798
400	Basic rotation using two points	■		565
401	Basic rotation over two holes	■		568
402	Basic rotation over two studs	■		572
403	Compensate misalignment with rotary axis	■		577
404	Set basic rotation	■		582
405	Compensate misalignment with the C axis	■		583
408	Preset at slot center (FCL 3 function)	■		593
409	Preset at ridge center (FCL 3 function)	■		597
410	Preset from inside of rectangle	■		601
411	Preset from outside of rectangle	■		605
412	Preset from inside of circle (hole)	■		609
413	Preset from outside of circle (stud)	■		614
414	Preset from outside of corner	■		619
415	Preset from inside of corner	■		624
416	Preset from circle center	■		629
417	Preset in touch probe axis	■		634
418	Preset at center between four holes	■		636
419	Preset in any one axis	■		641
420	Workpiece—measure angle	■		656
421	Workpiece—measure hole (center and diameter of hole)	■		659
422	Workpiece—measure circle from outside (diameter of circular stud)	■		664
423	Workpiece—measure rectangle from inside	■		669
424	Workpiece—measure rectangle from outside	■		672
425	Workpiece—measure inside width (slot)	■		675
426	Workpiece—measure outside width (ridge)	■		678
427	Workpiece—measure in any selectable axis	■		681
430	Workpiece—measure bolt hole circle	■		684

Cycle number	Cycle name	DEF active	CALL active	Page
431	Workpiece—measure plane	■		684
441	Fast probing	■		717
450	KinematicsOpt: Save kinematics (option)	■		748
451	KinematicsOpt: Measure kinematics (option)	■		751
452	KinematicsOpt: Preset compensation	■		744
453	Kinematics grid	■		776
460	Calibrate the touch probe	■		706
461	Calibrate touch probe length	■		710
462	Calibrate touch probe inside radius	■		712
463	Calibrate touch probe outside radius	■		714
480	Calibrate the TT	■		790
481	Measure/Inspect the tool length	■		794
482	Measure/Inspect the tool radius	■		796
483	Measure/Inspect the tool length and the tool radius	■		798
484	Calibrate TT	■		792
600	Global workspace	■		732
601	Local workspace	■		737
1410	Probing on edge	■		554
1411	Probing two circles	■		558
1420	Probing in plane	■		550

Index

R

S

T

U

W

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.de

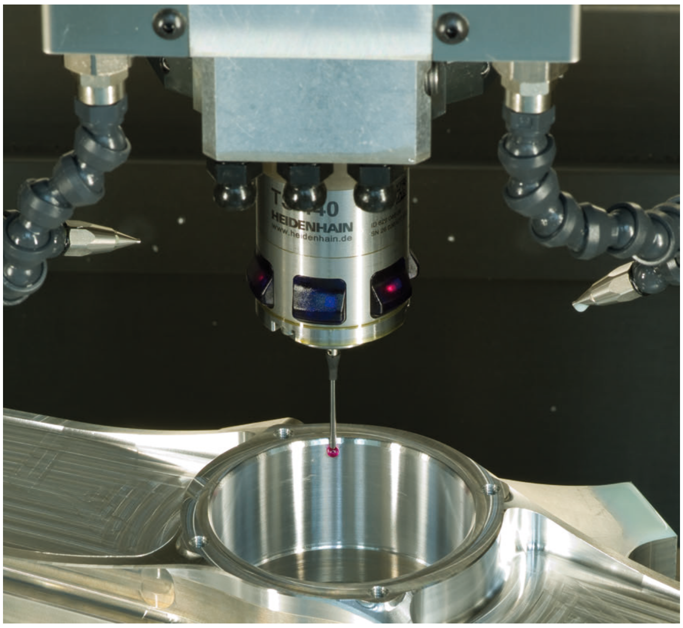
Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 220	Signal transmission by cable
TS 440, TS 444	Infrared transmission
TS 640, TS 740	Infrared transmission

- Workpiece alignment
- Setting presets
- Workpiece measurement



Tool touch probes

TT 140	Signal transmission by cable
TT 449	Infrared transmission
TL	Non-contacting laser systems

- Tool measurement
- wear monitoring
- tool breakage detection

Original document
892905-27 · Ver07 · SW09 · 10/2018 · H · Printed in Germany