



HEIDENHAIN

TNC 640

User's Manual
Programming of Measuring
Cycles for Workpieces and Tools

NC Software
34059x-18

English (en)
10/2023

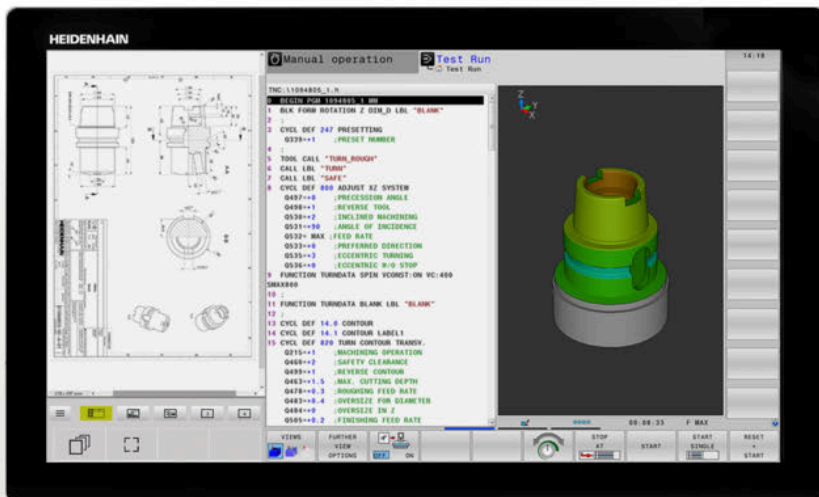


Table of contents

1	Fundamentals.....	19
2	Fundamentals / Overviews.....	33
3	Using Touch Probe Cycles.....	37
4	Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....	51
5	Touch Probe Cycles: Automatic Presetting.....	131
6	Touch Probe Cycles: Automatic Workpiece Inspection.....	233
7	Touch Probe Cycles: Special Functions.....	293
8	Touch Probe Cycles: Automatic Kinematics Measurement.....	331
9	Touch Probe Cycles: Automatic Tool Measurement.....	377
10	Cycles: Special Functions.....	411
11	Tables of Cycles.....	415

1	Fundamentals.....	19
1.1	About this manual.....	20
1.2	Control model, software, and features.....	22
	Software options.....	23
	New or modified cycle functions of software 34059x-18.....	29

2	Fundamentals / Overviews.....	33
2.1	Introduction.....	34
2.2	Available cycle groups.....	35
	Overview of machining cycles.....	35
	Overview of touch probe cycles.....	36

3	Using Touch Probe Cycles.....	37
3.1	General information about touch probe cycles.....	38
	Method of function.....	38
	Consideration of a basic rotation in the Manual operation mode.....	39
	Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes.....	39
	Touch probe cycles for automatic operation.....	40
3.2	Before you start working with touch probe cycles.....	42
	Maximum traverse to touch point: DIST in touch probe table.....	42
	Set-up clearance to touch point: SET_UP in touch probe table.....	42
	Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table.....	42
	Touch trigger probe, probing feed rate: F in touch probe table.....	43
	Touch trigger probe, rapid traverse for positioning: FMAX.....	43
	Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table.....	43
	Executing touch probe cycles.....	44
3.3	Program defaults for cycles.....	46
	Overview.....	46
	Entering GLOBAL DEF definitions.....	46
	Using GLOBAL DEF information.....	47
	Global data valid everywhere.....	48
	Global data for probing functions.....	49

4	Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....	51
4.1	Overview.....	52
4.2	Touch probe cycles 14xx: fundamentals.....	53
	Characteristics common to touch probe cycles 14xx for measuring rotations.....	53
	Semi-automatic mode.....	55
	Evaluation of tolerances.....	60
	Transferring the actual position.....	63
4.3	Cycle 1420 PROBING IN PLANE.....	64
	Cycle parameters.....	67
4.4	Cycle 1410 PROBING ON EDGE.....	71
	Cycle parameters.....	75
4.5	Cycle 1411 PROBING TWO CIRCLES.....	79
	Cycle parameters.....	83
4.6	Cycle 1412 INCLINED EDGE PROBING.....	87
	Cycle parameters.....	91
4.7	Cycle 1416 INTERSECTION PROBING.....	95
	Cycle parameters.....	99
4.8	Fundamentals of touch probe cycles 400 to 405.....	104
	Characteristics common to all touch probe cycles for measuring workpiece misalignment.....	104
4.9	Cycle 400 BASIC ROTATION.....	105
	Cycle parameters.....	107
4.10	Cycle 401 ROT OF 2 HOLES.....	109
	Cycle parameters.....	111
4.11	Cycle 402 ROT OF 2 STUDS.....	114
	Cycle parameters.....	116
4.12	Cycle 403 ROT IN ROTARY AXIS.....	119
	Cycle parameters.....	121
4.13	Cycle 405 ROT IN C AXIS.....	124
	Cycle parameters.....	126
4.14	Cycle 404 SET BASIC ROTATION.....	128
	Cycle parameters.....	128
4.15	Example: Determining a basic rotation from two holes.....	129

5	Touch Probe Cycles: Automatic Presetting	131
5.1	Overview	132
5.2	Fundamentals of touch probe cycles 14xx for preset setting	134
	Characteristics common to all touch probe cycles 14xx for preset setting	134
5.3	Cycle 1400 POSITION PROBING	135
	Cycle parameters	137
5.4	Cycle 1401 CIRCLE PROBING	139
	Cycle parameters	141
5.5	Cycle 1402 SPHERE PROBING	144
	Cycle parameters	146
5.6	Cycle 1404 PROBE SLOT/RIDGE	149
	Cycle parameters	151
5.7	Cycle 1430 PROBE POSITION OF UNDERCUT	153
	Cycle parameters	156
5.8	Cycle 1434 PROBE SLOT/RIDGE UNDERCUT	159
	Cycle parameters	162
5.9	Fundamentals of touch probe cycles 408 to 419 for preset setting	165
	Application	165
5.10	Cycle 410 PRESET INSIDE RECTAN.	167
	Cycle parameters	169
5.11	Cycle 411 PRESET OUTS. RECTAN.	172
	Cycle parameters	174
5.12	Cycle 412 PRESET INSIDE CIRCLE	178
	Cycle parameters	180
5.13	Cycle 413 PRESET OUTS. CIRCLE	184
	Cycle parameters	186
5.14	Cycle 414 PRESET OUTS. CORNER	190
	Cycle parameters	192
5.15	Cycle 415 PRESET INSIDE CORNER	196
	Cycle parameters	198
5.16	Cycle 416 PRESET CIRCLE CENTER	202
	Cycle parameters	204

5.17 Cycle 417 PRESET IN TS AXIS.....	208
Cycle parameters.....	210
5.18 Cycle 418 PRESET FROM 4 HOLES.....	212
Cycle parameters.....	214
5.19 Cycle 419 PRESET IN ONE AXIS.....	217
Cycle parameters.....	219
5.20 Cycle 408 SLOT CENTER PRESET.....	221
Cycle parameters.....	223
5.21 Cycle 409 RIDGE CENTER PRESET.....	226
Cycle parameters.....	228
5.22 Example: Presetting at center of a circular segment and on top surface of workpiece.....	231
5.23 Example: Presetting on top surface of workpiece and at center of a bolt hole circle.....	232

6	Touch Probe Cycles: Automatic Workpiece Inspection.....	233
6.1	Fundamentals.....	234
	Overview.....	234
	Recording the results of measurement.....	235
	Measurement results in Q parameters.....	237
	Classification of results.....	237
	Tolerance monitoring.....	237
	Tool monitoring.....	238
	Reference system for measurement results.....	239
6.2	Cycle 0 REF. PLANE.....	240
	Cycle parameters.....	241
6.3	Cycle 1 POLAR PRESET.....	242
	Cycle parameters.....	243
6.4	Cycle 420 MEASURE ANGLE.....	244
	Cycle parameters.....	245
6.5	Cycle 421 MEASURE HOLE.....	247
	Cycle parameters.....	249
6.6	Cycle 422 MEAS. CIRCLE OUTSIDE.....	253
	Cycle parameters.....	255
6.7	Cycle 423 MEAS. RECTAN. INSIDE.....	259
	Cycle parameters.....	261
6.8	Cycle 424 MEAS. RECTAN. OUTS.....	264
	Cycle parameters.....	265
6.9	Cycle 425 MEASURE INSIDE WIDTH.....	268
	Cycle parameters.....	269
6.10	Cycle 426 MEASURE RIDGE WIDTH.....	272
	Cycle parameters.....	273
6.11	Cycle 427 MEASURE COORDINATE.....	276
	Cycle parameters.....	278
6.12	Cycle 430 MEAS. BOLT HOLE CIRC.....	281
	Cycle parameters.....	283
6.13	Cycle 431 MEASURE PLANE.....	286
	Cycle parameters.....	288

6.14 Programming examples.....	290
Example: Measuring and reworking a rectangular stud.....	290
Example: Probing a rectangular pocket and recording the results.....	292

7	Touch Probe Cycles: Special Functions.....	293
7.1	Fundamentals.....	294
	Overview.....	294
7.2	Cycle 3 MEASURING.....	295
	Cycle parameters.....	296
7.3	Cycle 4 MEASURING IN 3-D.....	298
	Cycle parameters.....	299
7.4	Cycle 444 PROBING IN 3-D.....	300
	Cycle parameters.....	303
7.5	Cycle 441 FAST PROBING.....	306
	Cycle parameters.....	307
7.6	Cycle 1493 EXTRUSION PROBING.....	309
	Cycle parameters.....	313
7.7	Calibrating a touch trigger probe.....	314
7.8	Displaying calibration values.....	315
7.9	Cycle 461 TS CALIBRATION OF TOOL LENGTH.....	316
7.10	Cycle 462 CALIBRATION OF A TS IN A RING.....	318
7.11	Cycle 463 TS CALIBRATION ON STUD.....	321
7.12	Cycle 460 CALIBRATION OF TS ON A SPHERE.....	324

8	Touch Probe Cycles: Automatic Kinematics Measurement.....	331
8.1	Kinematics measurement with TS touch probes (option 48).....	332
	Fundamentals.....	332
	Overview.....	333
8.2	Requirements.....	334
	Notes.....	335
8.3	Cycle 450 SAVE KINEMATICS (option 48).....	336
	Cycle parameters.....	337
	Log function.....	338
	Notes on data management.....	338
8.4	Cycle 451 MEASURE KINEMATICS (option 48).....	339
	Positioning direction.....	342
	Machines with Hirth-coupled axes.....	343
	Example calculation of measuring positions for an A axis:.....	343
	Choice of number of measuring points.....	344
	Choice of the calibration sphere position on the machine table.....	344
	Notes on the accuracy.....	345
	Notes on various calibration methods.....	346
	Backlash.....	347
	Notes.....	348
	Cycle parameters.....	350
	Various modes (Q406).....	354
	Log function.....	356
8.5	Cycle 452 PRESET COMPENSATION (option 48).....	357
	Cycle parameters.....	362
	Adjustment of interchangeable heads.....	365
	Drift compensation.....	367
	Log function.....	369
8.6	Cycle 453 KINEMATICS GRID (option 48).....	370
	Various modes (Q406).....	371
	Choice of the calibration sphere position on the machine table.....	371
	Notes.....	372
	Cycle parameters.....	374
	Log function.....	376

9	Touch Probe Cycles: Automatic Tool Measurement.....	377
9.1	Fundamentals.....	378
	Overview.....	378
	Differences between Cycles 31 to 33 and Cycles 481 to 483.....	379
	Measuring a tool of length 0.....	380
	Setting machine parameters.....	381
	Entries in the tool table for milling and turning tools.....	383
9.2	Cycle 30 or 480 CALIBRATE TT.....	385
	Cycle parameters.....	387
9.3	Cycle 31 or 481 CAL. TOOL LENGTH.....	388
	Cycle parameters.....	390
9.4	Cycle 32 or 482 CAL. TOOL RADIUS.....	392
	Cycle parameters.....	395
9.5	Cycle 33 or 483 MEASURE TOOL.....	397
	Cycle parameters.....	400
9.6	Cycle 484 CALIBRATE IR TT.....	402
	Cycle parameters.....	405
9.7	Cycle 485 MEASURE LATHE TOOL (option 50 or 158).....	406
	Cycle parameters.....	410

10 Cycles: Special Functions.....	411
10.1 Fundamentals.....	412
Overview.....	412
10.2 Cycle 13 ORIENTATION.....	414
Cycle parameters.....	414

11 Tables of Cycles.....	415
11.1 Table of cycles.....	416
Touch probe cycles.....	416

1

Fundamentals

1.1 About this manual

Safety precautions

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

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

DANGER

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

WARNING

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

CAUTION

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

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in property damage**.

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.

In these instructions, you will find the following informational notes:



The information symbol indicates a **tip**.

A tip provides important additional or supplementary information.



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



The book symbol indicates a **cross reference**.

A cross reference leads to external documentation for example the documentation of your machine manufacturer or other supplier.

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

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

tnc-userdoc@heidenhain.de

1.2 Control model, software, and features

This manual describes programming functions provided by our controls with the following NC software numbers and later.

Control model	NC software number
TNC 640	340590-18
TNC 640 E	340591-18
TNC 640 programming station	340595-18

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

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

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

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

- Tool measurement with the TT

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

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



User's Manual:

All cycle functions not related to the measuring cycles are described in the **Programming of Machining Cycles** User's Manual. This manual is available from HEIDENHAIN upon request.

ID of User's Manual for Programming of Machining Cycles:
1303406-xx



User's Manual:

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.

Klartext 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, each of which can be enabled separately by your machine manufacturer. The respective options provide the functions listed below:

Additional Axis (option 0 to option 7)

Additional axis Additional control loops 1 to 8

Advanced Function Set 1 (option 8)

Advanced functions (set 1)

Machining with rotary tables

- Cylindrical contours as if in two axes
- Feed rate in distance per minute

Coordinate conversions:

Tilting the working plane

Interpolation:

Circular in 3 axes with tilted working plane

Advanced Function Set 2 (option 9)

Advanced functions (set 2)

Subject to export license

3D machining:

- 3D tool compensation through surface-normal vectors
- Changing the swivel-head angle with the electronic handwheel during program run without affecting the position of the tool tip (TCPM = **T**ool **C**enter **P**oint **M**anagement)
- 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 more than 4 axes (subject to export license)

HEIDENHAIN DNC (option 18)

Communication with external PC applications over COM component

DCM Collision (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, STL, STEP and IGES
- Adoption of contours and point patterns
- Simple and convenient specification of presets
- Selecting graphical features of contour sections from conversational programs

Global PGM Settings – GPS (option 44)

- Global program settings**
- Superimposition of coordinate transformations during program run
 - Handwheel superimpositioning
-

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
-

Turning (option 50)

- Milling and turning modes**
- Functions:**
- Switching between Milling/Turning mode of operation
 - Constant surface speed
 - Tool-tip radius compensation
 - Turning-specific contour elements
 - Turning cycles
 - Eccentric Turning
 - Cycle **880** or **G880 GEAR HOBGING** (options 50 and 131)
-

KinematicsComp (option 52)

- Three-dimensional compensation** Compensation of position and component errors
-

OPC UA NC Server (1 to 6) (options 56 to 61)

- Standardized interface**
- The OPC UA NC Server provides a standardized interface (**OPC UA**) for external access to the control's data and functions.
- These software options allow you to create up to six parallel client connections.
-

3D-ToolComp (option 92)

- 3D tool radius compensation depending on the tool's contact angle**
- Subject to export license
- 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 option 9)
-

Extended Tool Management (option 93)

Extended tool management	Python-based expansion of tool management <ul style="list-style-type: none"> ■ Program-specific or pallet-specific usage sequence of all tools ■ Program-specific or pallet-specific tooling list of all tools
---------------------------------	--

Advanced Spindle Interpolation (option 96)

Interpolating spindle	Interpolation turning: <ul style="list-style-type: none"> ■ Cycle 291 COUPLG.TURNG.INTERP. (ISO: G291) ■ Cycle 292 CONTOUR.TURNG.INTRP. (ISO: G292)
------------------------------	--

Spindle Synchronism (option 131)

Spindle synchronization	<ul style="list-style-type: none"> ■ Synchronization of milling spindle and turning spindle ■ Cycle 880 GEAR HOBGING (ISO: G880) (options 50 and 131)
--------------------------------	---

Remote Desktop Manager (option 133)

Remote operation of external computer units	<ul style="list-style-type: none"> ■ 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
----------------------------------	--

Cross Talk Compensation – CTC (option 141)

Compensation of axis couplings	<ul style="list-style-type: none"> ■ Determination of dynamically caused position deviation through axis acceleration ■ Compensation of the TCP (Tool Center Point)
---------------------------------------	--

Position Adaptive Control – PAC (option 142)

Adaptive position control	<ul style="list-style-type: none"> ■ Adaptation of the control parameters depending on the position of the axes in the working space ■ Adaptation of the control parameters depending on the speed or acceleration of an axis
----------------------------------	---

Load Adaptive Control – LAC (option 143)

Adaptive load control	<ul style="list-style-type: none"> ■ Automatic determination of workpiece weight and frictional forces ■ Adaptation of the control parameters depending on the current mass of the workpiece
------------------------------	--

Active Chatter Control – ACC (option 145)

Active chatter control	Fully automatic function for chatter control during machining
-------------------------------	---

Machine Vibration Control – MVC (option 146)

Vibration damping for machines	Damping of machine oscillations for improving the workpiece surface quality through the following functions: <ul style="list-style-type: none"> ■ Active Vibration Damping (AVD) ■ Frequency Shaping Control (FSC)
---------------------------------------	--

CAD Model Optimizer (option 152)

Optimization of CAD models	Conversion and optimization of CAD models <ul style="list-style-type: none"> ■ Fixtures ■ Workpiece blank ■ Finished part
-----------------------------------	--

Batch Process Manager (option 154)

Batch process manager	Planning of production orders
------------------------------	-------------------------------

Component Monitoring (option 155)

Component monitoring without external sensors	Monitoring of configured machine components for overload
--	--

Grinding (option 156)

Jig grinding	<ul style="list-style-type: none"> ■ Reciprocating stroke cycles ■ Cycles for dressing ■ Support of the "dressing tool" and "grinding tool" tool types
---------------------	---

Gear Cutting (option 157)

Machining gear systems	<ul style="list-style-type: none"> ■ Cycle 285 DEFINE GEAR (ISO: G285) ■ Cycle 286 GEAR HOBGING (ISO: G286) ■ Cycle 287 GEAR SKIVING (ISO: G287)
-------------------------------	---

Turning v2 (option 158)

Mill-turning version 2	<ul style="list-style-type: none"> ■ All functions of software option 50 ■ Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING ■ Cycle 883 TURNING SIMULTANEOUS FINISHING <p>The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation.</p>
-------------------------------	---

Opt. contour milling (option 167)

Optimized contour cycles	Cycles for machining any pockets and islands using trochoidal milling
---------------------------------	---

Further options available


HEIDENHAIN offers further hardware enhancements and software options that can be configured and implemented only by your machine manufacturer. This includes functional safety (FS), for example.

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

ID: 827222-xx

**VTC User's Manual**

All functions of the software for the VT 121 vision system are described in the **VTC User's Manual**. Please contact HEIDENHAIN if you require a copy of this User's Manual.
ID: 1322445-xx

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 manufacturer 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**Legal information**

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

Further information is available on the control as follows:

- ▶ Press the **MOD** key to open the **Settings and information** dialog
- ▶ Select **Code-number entry** in the dialog
- ▶ Press the **LICENSE INFO** soft key or select **Settings and information, General information** → **License info** directly in the dialog

Furthermore, the control software contains binary libraries of the **OPC UA** software from Softing Industrial Automation GmbH. For these libraries, the terms of use agreed upon between HEIDENHAIN and Softing Industrial Automation GmbH shall additionally apply and prevail.

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

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 modified cycle functions of software 34059x-18 " 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.

Proceed as follows:

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

Compatibility

Most NC programs created with older HEIDENHAIN contouring controls (with TNC 150 B and later) 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 downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

New or modified cycle functions of software 34059x-18



Overview of new and modified software functions

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

ID: 1322095-xx

New cycle functions with 81762x-18

- Cycle **1274 OCM CIRCULAR SLOT** (ISO: **G1274**, option 167)
This cycle allows you to define a circular slot that is then used as a pocket or boundary for face milling in conjunction with other OCM cycles.

Modified cycle functions with 81762x-18

- You can also define subcontours as **LBL** subprograms within the complex **SEL CONTOUR** contour formula.
- The machine manufacturer can hide the cycles **220 POLAR PATTERN** (ISO: **G220**) and **221 CARTESIAN PATTERN** (ISO: **G221**). We recommend using the **PATTERN DEF** function.
- The input value **1** has been added to parameter **Q515 FONT** in Cycle **225 ENGRAVING** (ISO: **G225**). Use this input value to select the **LiberationSans-Regular** font.
- In the following cycles, you can enter symmetrical tolerances "+-...." for the nominal dimensions:
 - Cycle **208 BORE MILLING** (ISO: **G208**)
 - **127x** (option 167) OCM standard figure cycles
- Cycle **287 GEAR SKIVING** (ISO: **G287**, option 157) has been extended:
 - When you program the optional parameter **Q466 OVERRUN PATH**, the control will optimize the approach and idle travel paths automatically. This will reduce machining times.
 - Two columns have been added to the prototype of the technology table:
 - **dk**: Angular offset of the workpiece in order to machine one side of the tooth flank only. This can be used to increase the surface quality.
 - **PGM**: Profile program for a custom tooth flank line, for example to realize crowning of the tooth flank.
 - After each step, the control displays the number of the current cut and the number of remaining cuts in a pop-up window.
- The machine manufacturer can configure a deviating automatic **LIFTOFF** for Cycles **286 GEAR HOBBING** (ISO: **G286**, option 157) and **287 GEAR SKIVING** (ISO: **G287**, option 157).
- Cycle **800 ADJUST XZ SYSTEM** (ISO: **G800**, option 50) has been extended:
 - The input range of the parameter **Q497 PRECESSION ANGLE** has been extended from four to five decimal places.
 - The input range of the parameter **Q531 ANGLE OF INCIDENCE** has been extended from three to five decimal places.
- The control shows remaining residual material during turning cycles also with the machining operations **Q215=1** and **Q215=2**.

- In the touch probe cycles **14xx**, you can enter symmetrical tolerances "+-..." for the nominal dimensions.
Further information: "Evaluation of tolerances", Page 60
- Cycle **441 FAST PROBING** (ISO: **G441**) now features the parameter **Q371 TOUCH POINT REACTION**. This parameter defines the reaction of the control in cases where the stylus is not deflected.
Further information: "Cycle 441 FAST PROBING", Page 306
- Using the parameter **Q400 INTERRUPTION** in Cycle **441 FAST PROBING** (ISO: **G441**), you can define whether the control will interrupt program run and display a measuring log. The parameter is effective in conjunction with the following cycles:
 - Cycle **444 PROBING IN 3-D** (ISO: **G444**)
 - Touch-probe cycles **45x** for kinematics measuring
 - Touch-probe cycles **46x** for calibrating the workpiece touch probe
 - Touch-probe cycles **14xx** for determining a workpiece misalignment and for acquiring the preset**Further information:** "Cycle 441 FAST PROBING", Page 306
- Cycles **451 MEASURE KINEMATICS** (ISO: **G451**, option 48) and **452 PRESET COMPENSATION** (ISO: **452**, option 48) save the measured position errors of the rotary axes in the QS parameters **QS144** to **QS146**.
Further information: "Cycle 451 MEASURE KINEMATICS (option 48)", Page 339
Further information: "Cycle 452 PRESET COMPENSATION (option 48)", Page 357
- Using the optional machine parameter **maxToolLengthTT** (no. 122607), the machine manufacturer defines a maximum tool length for tool touch probe cycles.
Further information: "Measuring a tool of length 0", Page 380
- Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.
Further information: "Setting machine parameters", Page 381

2

**Fundamentals /
Overviews**

2.1 Introduction



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

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

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

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

- ▶ Test your program before executing it



If you use indirect parameter assignments in cycles with numbers greater than **200** (e.g., **Q210 = Q1**), any change in the assigned 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 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 machining cycles











- ▶ Press the **CYCL DEF** key

Soft key	Cycle group	Page
	Cycles for pecking, reaming, boring and counterboring	Further information: User's Manual for Programming of Machining Cycles
	Cycles for tapping, thread cutting and thread milling	Further information: User's Manual for Programming of Machining Cycles
	Cycles for milling pockets, studs, slots, and face milling	Further information: User's Manual for Programming of Machining Cycles
	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	Further information: User's Manual for Programming of Machining 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	Further information: User's Manual for Programming of Machining Cycles
	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	Further information: User's Manual for Programming of Machining Cycles
	Cycles for turning and gear hobbing	Further information: User's Manual for Programming of Machining Cycles
	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning, determining the load, gear cycles	Further information: User's Manual for Programming of Machining Cycles
	Cycles for grinding operations and sharpening of grinding tools	Further information: User's Manual for Programming of Machining Cycles
	<ul style="list-style-type: none"> ▶ If required, switch to machine-specific machining cycles <p>The machine manufacturer can integrate these types of machining cycles.</p>	

Overview of touch probe cycles



- ▶ Press the **TOUCH PROBE** key.

Soft key	Cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	52
	Cycles for automatic workpiece presetting	132
	Cycles for automatic workpiece inspection	234
	Special cycles	294
	Touch probe calibration	314
	Cycles for automatic kinematics measurement	333
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	378
	▶ Switch to machine-specific touch probe cycles, if available; these touch probe cycles can be integrated by the machine manufacturer	

3

**Using Touch Probe
Cycles**

3.1 General information about touch probe cycles



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

While touch probe functions are being executed, the control temporarily disables the **Global Program Settings**.

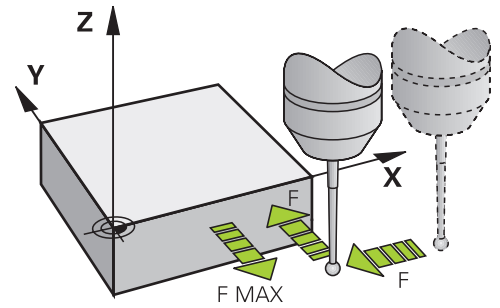


HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

Method of function



- Refer to your machine manual.
- The control must be specifically prepared by the machine manufacturer for the use of a 3D touch probe.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- The control's full range of functions is available only if the **Z** tool axis is used.
- Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.



Whenever the control runs a touch probe cycle, the 3D touch probe approaches the workpiece parallel to the axis. This is also true during an active basic rotation or with a tilted working plane. The machine manufacturer will determine the probing feed rate in a machine parameter.

Further information: "Before you start working with touch probe cycles", Page 42

When the probe stylus contacts the workpiece,

- the 3D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the control displays an error message (distance: **DIST** from touch probe table).

Requirements

- Calibrated workpiece touch probe

Further information: "Calibrating a touch trigger probe", Page 314

Working with an L-shaped stylus

In addition to a **SIMPLE** stylus, probing cycles **444** and **14xx** also support the **L-TYPE** stylus, which is L-shaped. The L-shaped stylus must be calibrated prior to use.

HEIDENHAIN recommends calibrating the stylus with the following cycles:

- Radius calibration: Cycle 460 CALIBRATION OF TS ON A SPHERE
- Length calibration: Cycle 461 TS CALIBRATION OF TOOL LENGTH

Stylus orientation must be permitted via **TRACK ON** in the touch probe table. During the probing process, the control orients the L-shaped stylus to the given probing direction. If the probing direction is identical to the tool axis, then the control orients the touch probe to the calibration angle.



- The control does not show the arm of the stylus in the simulation. The arm is the angled part of the L-shaped stylus.
- In order to achieve maximum accuracy, the feed rate during calibration must be identical to the feed rate during probing.

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

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** operating modes, the control provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensate workpiece misalignment
- Set presets

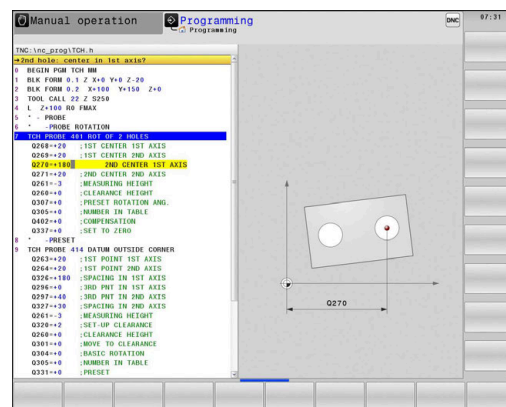
Touch probe cycles for automatic operation

Besides the touch probe cycles, which you can use in the Manual Operation and Electronic handwheel operating 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 machining cycles, touch probe cycles with numbers greater than **400** use Q parameters as transfer parameters. Parameters with the same function that the control requires in several cycles always have the same number: For example, **Q260** is always assigned the clearance height, **Q261** the measuring height, etc.

To simplify programming, the control shows a graphic during cycle definition. In the graphic, the parameter that needs to be entered is highlighted (see figure right).



Defining the touch probe cycle in the Programming mode of operation

Proceed as follows:



- ▶ Press the **TOUCH PROBE** key.



- ▶ Select the desired measuring cycle group (e.g., presetting)
- ▶ Cycles for automatic tool measurement are available only if your machine has been prepared for them.



- ▶ Select the cycle (e.g., **PRESET INSIDE RECTAN**)
- ▶ The control opens the programming dialog and prompts you for all required input values. At the same time, a graphic of the input parameters is displayed in the right screen half. The parameter you are prompted for in the dialog is highlighted.
- ▶ Enter all parameters required by the control
- ▶ Confirm each input with the **ENT** key
- ▶ The control closes the dialog when all required data has been entered.

NC blocks

11 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

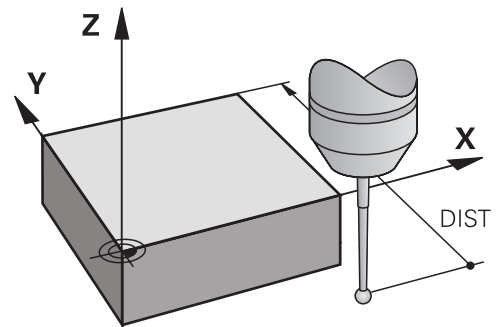
3.2 Before you start working with touch probe cycles

To make it possible to cover the widest possible range of measuring tasks, you have various possibilities for defining the behavior common to all touch probe cycles.

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

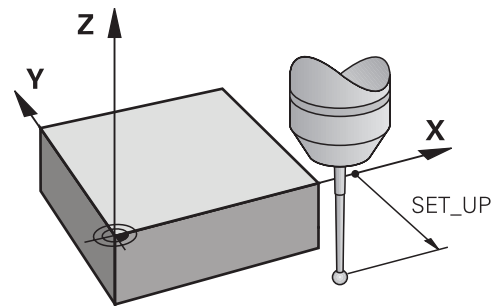
Maximum traverse to touch point: **DIST** in touch probe table

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



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

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



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

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



If you change **TRACK = ON**, you must recalibrate the touch probe.

Touch trigger probe, probing feed rate: **F** in touch probe table

In **F**, you define the feed rate at which the control is to probe the workpiece.

F can never be greater than defined in the optional machine parameter **maxTouchFeed** (no. 122602).

The feed-rate potentiometer can be effective in touch probe cycles. The machine manufacturer enters the necessary settings. (Parameter **overrideForMeasure** (no. 122604), must be configured correspondingly.)

Touch trigger probe, rapid traverse for positioning: **FMAX**

In **FMAX**, you define the feed rate at which the control pre-positions the touch probe and positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: **F_PREPOS** in touch probe table

In **F_PREPOS**, you define whether the control is to position the touch probe at the feed rate defined in **FMAX** or at rapid traverse.

- Input value = **FMAX_PROBE**: Position at feed rate from **FMAX**
- Input value = **FMAX_MACHINE**: Pre-position at rapid traverse

Executing touch probe cycles

All touch probe cycles are DEF-active. The control runs the cycle automatically as soon as it reads the cycle definition in the program run.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

Note regarding machine parameters

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

Notes in connection with programming and execution

- Please note that the units of measure in the measuring log and in return parameters depend on the setting in the main program.
- The touch probe cycles **40x** to **43x** will reset an active basic rotation at the beginning of the cycle.
- The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.
- You can apply the inclined position as a workpiece rotation only if a table rotary axis exists on the machine and if its orientation is perpendicular to the workpiece coordinate system **W-CS**.

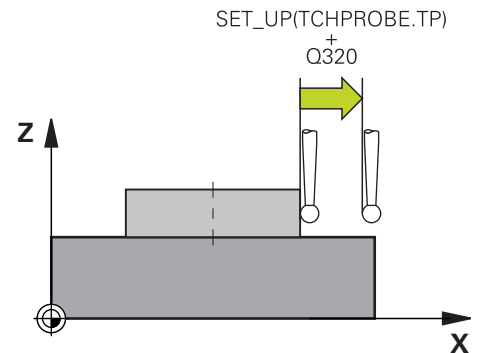
Pre-positioning

Before each probing operation, the control pre-positions the touch probe.

Pre-positioning is done in the inverse probing direction.

The distance between the probing point and the pre-position results from the following values:

- Ball-tip radius **R**
- **SET_UP** from the touch-probe table
- **Q320 SET-UP CLEARANCE**



Positioning logic

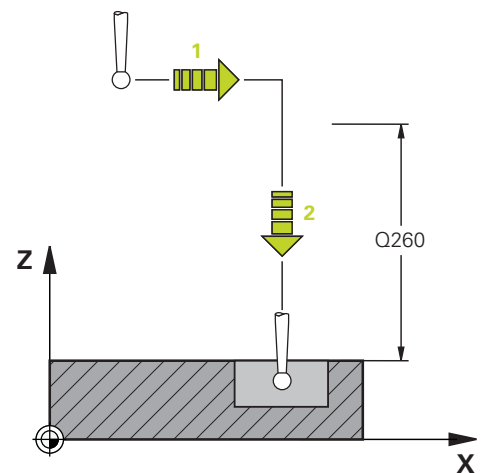
Touch-probe cycles with numbers from **400** through **499** or **1400** through **1499** pre-position the touch probe according to the following positioning logic:

Current position > Q260 CLEARANCE HEIGHT

- 1 The control positions the touch probe at **FMAX** to the pre-position in the working plane.

Further information: "Pre-positioning ", Page 45

- 2 Then, the control positions the touch probe at **FMAX** in the tool axis, directly to the probing height.



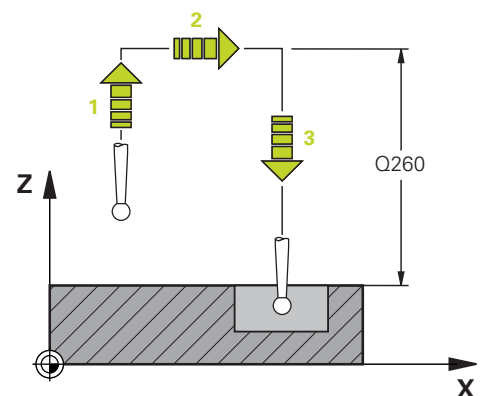
Current position < Q260 CLEARANCE HEIGHT

- 1 The control positions the touch probe at **FMAX** to **Q260 CLEARANCE HEIGHT**.

- 2 The control positions the touch probe at **FMAX** to the pre-position in the working plane.

Further information: "Pre-positioning ", Page 45

- 3 Then, the control positions the touch probe at **FMAX** in the tool axis, directly to the probing height.

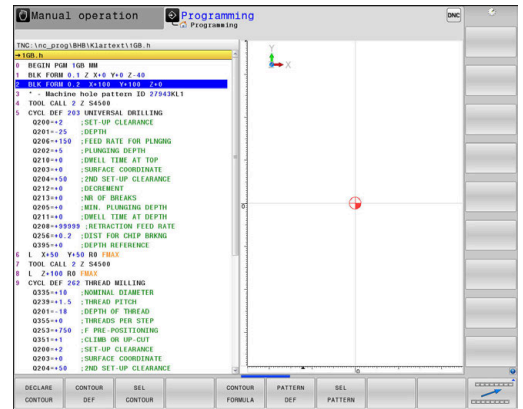


3.3 Program defaults for cycles

Overview

Some cycles always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** function you can define these cycle parameters at the beginning of the program, so that they are effective globally for all cycles used in the NC program. In the respective cycle you then simply reference the value defined at the beginning of the program.

The following **GLOBAL DEF** functions are available:

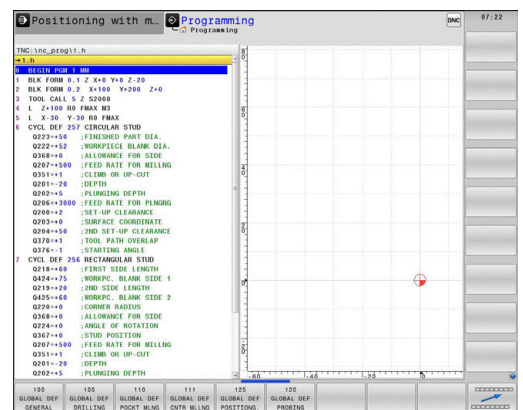


Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF GENERAL Definition of generally valid cycle parameters	48
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	Further information: User's Manual for Programming of Machining Cycles
110 GLOBAL DEF POCKET MILLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	Further information: User's Manual for Programming of Machining Cycles
111 GLOBAL DEF CNTR MILLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	Further information: User's Manual for Programming of Machining Cycles
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior with CYCL CALL PAT	Further information: User's Manual for Programming of Machining Cycles
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	49

Entering GLOBAL DEF definitions

Proceed as follows:






- ▶ Press the **PROGRAMMING** key
- ▶ Press the **SPEC FCT** key
- ▶ Press the **PROGRAM DEFAULTS** soft key
- ▶ Press the **GLOBAL DEF** soft key
- ▶ Select the desired GLOBAL DEF function (e.g., by pressing the **GLOBAL DEF PROBING** soft key)
- ▶ Enter the required definitions
- ▶ Press the **ENT** key each time to confirm

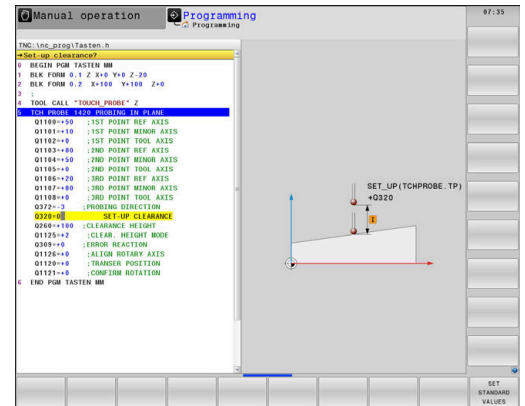


Using GLOBAL DEF information

If you entered the appropriate GLOBAL DEF functions at the beginning of the program, then you can reference these globally effective values when defining any cycles.

Proceed as follows:

-  ▶ Press the **PROGRAMMING** key
-  ▶ Press the **TOUCH PROBE** key.
-  ▶ Select the desired cycle group (e.g., Rotation)
-  ▶ Select the desired cycle (e.g., **PROBING IN PLANE**)
 - If a global parameter exists, the control will display the **SET STANDARD VALUES** soft key.
-  ▶ Press the **SET STANDARD VALUES** soft key
 - The control enters the word **PREDEF** in the cycle definition. This creates a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program.



NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly. There is a danger of collision!

- ▶ Make sure to use **GLOBAL DEF** carefully. Test your program before executing it
- ▶ If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

Global data valid everywhere

The parameters are valid for all **2xx** machining cycles as well as for Cycles **880, 1017, 1018, 1021, 1022, 1025** and touch probe cycles **451, 452, 453**

Help graphic	Parameter
	<p>Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: 0...99999.9999</p>
	<p>Q204 2nd set-up clearance? Distance in the tool axis between the tool and the workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: 0...99999.9999</p>
	<p>Q253 Feed rate for pre-positioning? Feed rate at which the control moves the tool within a cycle. Input: 0...99999.999 or FMAX, FAUTO</p>
	<p>Q208 Feed rate for retraction? Feed rate at which the control retracts the tool. Input: 0...99999.999 or FMAX, FAUTO</p>

Example

11 GLOBAL DEF 100 GENERAL ~	
Q200=+2	;SET-UP CLEARANCE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q253=+750	;F PRE-POSITIONING ~
Q208=+999	;RETRACTION FEED RATE

Global data for probing functions

The parameters apply to all touch-probe cycles **4xx** and **14xx** as well as the Cycles **271, 286, 287, 880, 1021, 1022, 1025, 1271, 1272, 1273, 1274, 1278**

Help graphic	Parameter
	<p>Q320 Set-up clearance?</p> <p>Additional distance between touch point and ball tip. Q320 is active in addition to the SET_UP column in the touch probe table. This value has an incremental effect.</p> <p>Input: 0...99999.9999 or PREDEF</p>
	<p>Q260 Clearance height?</p> <p>Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999 or PREDEF</p>
	<p>Q301 Move to clearance height (0/1)?</p> <p>Define how the touch probe will move between the measuring points:</p> <p>0: Move to measuring height between measuring points 1: Move to clearance height between measuring points</p> <p>Input: 0, 1</p>

Example

11 GLOBAL DEF 120 PROBING ~	
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q301=+1	;MOVE TO CLEARANCE

4

**Touch Probe
Cycles: Automatic
Measurement
of Workpiece
Misalignment**

4.1 Overview



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

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

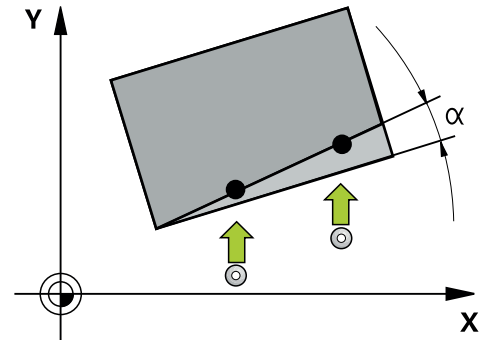
Soft key	Cycle	Page
	Cycle 1420 PROBING IN PLANE <ul style="list-style-type: none"> ■ Automatic measurement using three points ■ Compensation via basic rotation or rotary table rotation 	64
	Cycle 1410 PROBING ON EDGE <ul style="list-style-type: none"> ■ Automatic measurement using two points ■ Compensation via basic rotation or rotary table rotation 	71
	Cycle 1411 PROBING TWO CIRCLES <ul style="list-style-type: none"> ■ Automatic measurement using two holes or studs ■ Compensation via basic rotation or rotary table rotation 	79
	Cycle 1412 INCLINED EDGE PROBING <ul style="list-style-type: none"> ■ Automatic measurement using two points on an inclined edge ■ Compensation via basic rotation or rotary table rotation 	87
	Cycle 1416 INTERSECTION PROBING <ul style="list-style-type: none"> ■ Automatically determines the intersection with four touch points on two straight lines ■ Compensation via basic rotation or rotary table rotation 	95
	Cycle 400 BASIC ROTATION <ul style="list-style-type: none"> ■ Automatic measurement using two points ■ Compensation via basic rotation 	105
	Cycle 401 ROT OF 2 HOLES <ul style="list-style-type: none"> ■ Automatic measurement using two holes ■ Compensation via basic rotation 	109
	Cycle 402 ROT OF 2 STUDS <ul style="list-style-type: none"> ■ Automatic measurement using two studs ■ Compensation via basic rotation 	114
	Cycle 403 ROT IN ROTARY AXIS <ul style="list-style-type: none"> ■ Automatic measurement using two points ■ Compensation via rotary table rotation 	119
	Cycle 405 ROT IN C AXIS <ul style="list-style-type: none"> ■ Automatic alignment of an angular offset between a hole center and the positive Y axis ■ Compensation via rotary table rotation 	124
	Cycle 404 SET BASIC ROTATION <ul style="list-style-type: none"> ■ Setting any basic rotation 	128

4.2 Touch probe cycles 14xx: fundamentals

Characteristics common to touch probe cycles 14xx for measuring rotations

These cycles can determine rotations. They contain the following:

- Consideration of active machine kinematics
- Semi-automatic probing
- Monitoring of tolerances
- Consideration of 3D calibration
- Simultaneous measurement of rotation and position



Programming and operating notes:

- The probing positions are referenced to the programmed nominal coordinates in the I-CS.
- See the drawing for these nominal positions.
- Before a cycle definition you must program a tool call to define the touch-probe axis.
- The 14xx probing cycles support the **SIMPLE** and **L-TYPE** styli.
- In order to achieve optimal accuracy results with an L-TYPE stylus, HEIDENHAIN recommends that you perform probing and calibration at the same speed. Note the setting of the feed override if it is active during probing.

Explanation of terms

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

Evaluation – preset:

- If you want to probe objects in a consistent machining plane or probe objects while TCPM is active, you can program any required shifts as basic transformations in the preset table.
- Rotations can be written to the basic transformations of the preset table as basic rotations or as axial offsets from the first rotary table axis, seen from the workpiece.

**Operating notes:**

- When probing, existing 3D calibration data are taken into account. If these calibration data do not exist, deviations might be the result.
- If you wish to use not only the rotation but also a measured position, then probe as perpendicularly to the surface as possible. The larger the angular error and the larger the ball-tip radius, the larger the positioning error. If the angular errors in the initial angular position are too large, corresponding position errors might be the result.

Logging:

The measured results are recorded in the **TCHPRAUTO.html** file and stored in the Q parameters programmed for this cycle.

The measured deviations are the differences between the measured actual values and the mean tolerance value. If no tolerance has been specified, they refer to the nominal dimension.

The unit of measurement of the main program can be seen in the header of the log.

Semi-automatic mode

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

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

Cycle sequence:

- 1 The cycle interrupts the NC program.
- 2 A dialog window opens

Proceed as follows:

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

NOTICE

Danger of collision!

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

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



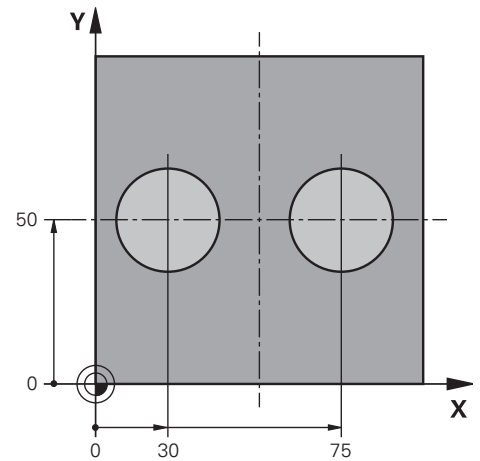
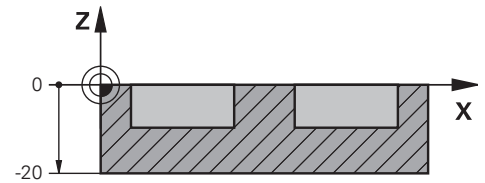
Programming and operating notes:

- See the drawing for the nominal positions.
- Semi-automatic mode can only be executed in the machine operating modes, i.e. not in the Test run operating mode.
- If you did not define a nominal position for a touch point in any direction, the control generates an error message.
- If you did not define a nominal position for a single direction, the control will capture the actual position after probing the object. This means that the measured actual position will subsequently be applied as the nominal position. Consequentially, there is no deviation for this position and thus no position compensation.

Examples

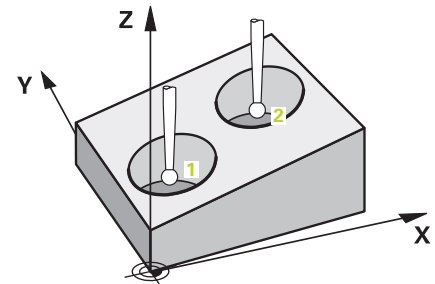
Important: Specify the **nominal positions** from the drawing!

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



Hole

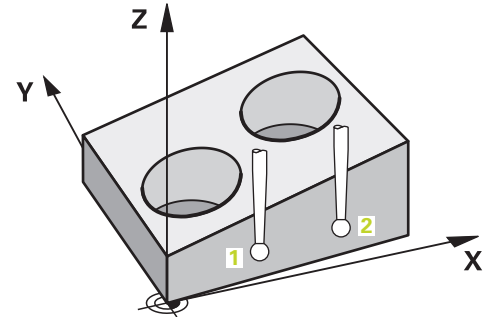
In this example, you will align two holes. Probing is done in the X axis (main axis) and in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure in this direction.



11 TCH PROBE 1411 PROBING TWO CIRCLES ~		; Define the cycle
QS1100= "?30" ;1ST POINT REF AXIS ~		; Nominal Position 1 of the main axis is provided, but the workpiece position is not known
QS1101= "?50" ;1ST POINT MINOR AXIS ~		; Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
QS1102= "?" ;1ST POINT TOOL AXIS ~		; Nominal position 1 in tool axis is unknown
Q1116=+10 ;DIAMETER 1 ~		; Diameter at position 1
QS1103= "?75" ;2ND POINT REF AXIS ~		; Nominal Position 2 of the main axis is provided, but the workpiece position is not known
QS1104= "?50" ;2ND POINT MINOR AXIS ~		; Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
QS1105= "?" ;2ND POINT TOOL AXIS ~		; Nominal position 2 in tool axis is unknown
Q1117=+10 ;DIAMETER 2 ~		; Diameter at position 2
Q1115=+0 ;GEOMETRY TYPE ~		; Geometry type: two holes
Q423=+4 ;NO. OF PROBE POINTS ~		
Q325=+0 ;STARTING ANGLE ~		
Q1119=+360 ;ANGULAR LENGTH ~		
Q320=+2 ;SET-UP CLEARANCE ~		
Q260=+100 ;CLEARANCE HEIGHT ~		
Q1125=+2 ;CLEAR. HEIGHT MODE ~		
Q309=+0 ;ERROR REACTION ~		
Q1126=+0 ;ALIGN ROTARY AXIS ~		
Q1120=+0 ;TRANSER POSITION ~		
Q1121=+0 ;CONFIRM ROTATION		

Edge

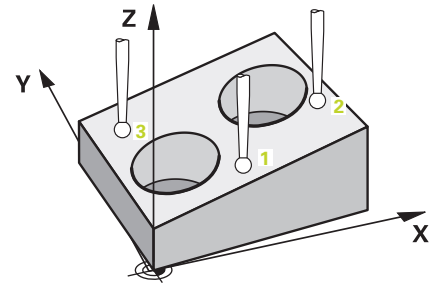
In this example, you will align an edge. Probing is done in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for this axis! Nominal positions for the X axis (main axis) and for the Z axis (tool axis) are not required because you will not measure in these directions.



11 TCH PROBE 1410 PROBING ON EDGE ~		; Define the cycle ~
QS1100= "?"	;1ST POINT REF AXIS ~	; Nominal position 1 in main axis is unknown
QS1101= "?0"	;1ST POINT MINOR AXIS ~	; Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
QS1102= "?"	;1ST POINT TOOL AXIS ~	; Nominal position 1 in tool axis is unknown
QS1103= "?"	;2ND POINT REF AXIS ~	; Nominal position 2 in main axis is unknown
QS1104= "?0"	;2ND POINT MINOR AXIS ~	; Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
QS1105= "?"	;2ND POINT TOOL AXIS ~	; Nominal position 2 in tool axis is unknown
Q372=+2	;PROBING DIRECTION ~	; Probing direction Y+
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	

Plane

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



11 TCH PROBE 1420 PROBING IN PLANE ~		; Define the cycle
QS1100= "?50" ;1ST POINT REF AXIS ~		; Nominal Position 1 of the main axis is provided, but the workpiece position is not known
QS1101= "?10" ;1ST POINT MINOR AXIS ~		; Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
QS1102= "?0" ;1ST POINT TOOL AXIS ~		; Nominal Position 1 of the tool axis is provided, but the workpiece position is not known
QS1103= "?80" ;2ND POINT REF AXIS ~		; Nominal Position 2 of the main axis is provided, but the workpiece position is not known
QS1104= "?50" ;2ND POINT MINOR AXIS ~		; Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
QS1105= "?0" ;2ND POINT TOOL AXIS ~		; Nominal Position 2 of the tool axis is provided, but the workpiece position is not known
QS1106= "?20" ;3RD POINT REF AXIS ~		; Nominal Position 3 of the main axis is provided, but the workpiece position is not known
QS1107= "?80" ;3RD POINT MINOR AXIS ~		; Nominal Position 3 of the secondary axis is provided, but the workpiece position is not known
QS1108= "?0" ;3RD POINT TOOL AXIS ~		; Nominal Position 3 of the tool axis is provided, but the workpiece position is not known
Q372=-3 ;PROBING DIRECTION ~		; Probing direction Z-
Q320=+2 ;SET-UP CLEARANCE ~		
Q260=+100 ;CLEARANCE HEIGHT ~		
Q1125=+2 ;CLEAR. HEIGHT MODE ~		
Q309=+0 ;ERROR REACTION ~		
Q1126=+0 ;ALIGN ROTARY AXIS ~		
Q1120=+0 ;TRANSER POSITION ~		
Q1121=+0 ;CONFIRM ROTATION		

Evaluation of tolerances

Cycles 14xx also allow you to check tolerance bands. This includes the checking of the position and size of an object.

The following input values with tolerances are possible:

Tolerances	Example
DIN EN ISO 286-2	10H7
ISO 2768-1	10 m
Dimensions	10+0.01-0.015

For the deviations, the following combinations are possible:

Combination	Example	Manufacturing dimension
x+y	10+-0.5	10.0
x-y	10-+0.5	10.0
x-y+z	10-0.1+0.5	10.2
x+y-z	10+0.1-0.5	9.8
x+y+z	10+0.1+0.5	10.3
x-y-z	10-0.1-0.5	9.7
x+y	10+0.5	10.25
x-y	10-0.5	9.75

If you enter a value with a tolerance in your program, the control will monitor the tolerance band. The control writes the following statuses to the return parameter **Q183**: Good, rework, or scrap. If a correction of the preset is programmed, the control corrects the active preset after probing.

The following cycle parameters allow input values with tolerances:

- **Q1100 1ST POINT REF AXIS**
- **Q1101 1ST POINT MINOR AXIS**
- **Q1102 1ST POINT TOOL AXIS**
- **Q1103 2ND POINT REF AXIS**
- **Q1104 2ND POINT MINOR AXIS**
- **Q1105 2ND POINT TOOL AXIS**
- **Q1106 3RD POINT REF AXIS**
- **Q1107 3RD POINT MINOR AXIS**
- **Q1108 3RD POINT TOOL AXIS**
- **Q1116 DIAMETER 1**
- **Q1117 DIAMETER 2**

To program this:

- ▶ Start the cycle definition
- ▶ Define the cycle parameters
- ▶ Press the **ENTER TEXT** soft key
- ▶ Enter a nominal dimension including tolerance



If you program an incorrect tolerance, the control will interrupt machining with an error message.

Cycle sequence

If the actual position is outside the tolerance, the control behaves as follows:

- **Q309 = 0**: The control does not interrupt program run.
- **Q309 = 1**: In the case of scrap or rework, the control interrupts program run with a message.
- **Q309 = 2**: In the case of scrap, the control interrupts program run with a message.

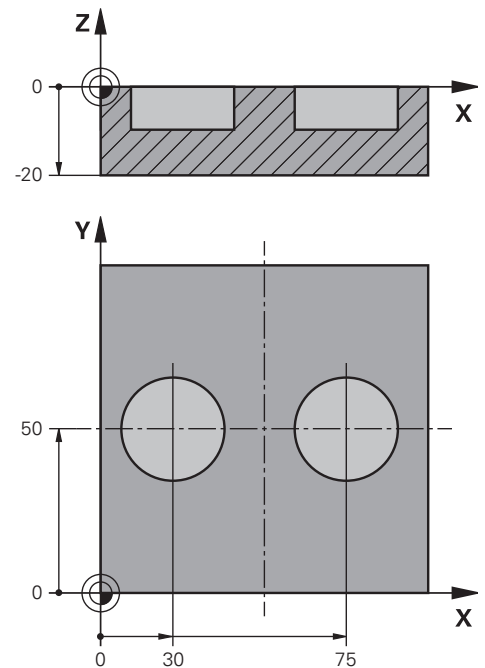
If Q309 = 1 or 2:

- The control opens a dialog and displays all of the nominal and actual dimensions of the object.
- ▶ Press the **CANCEL** soft key to interrupt NC program run
- or
- ▶ Press **NC Start** to resume NC program run



Please note that the deviations returned by the touch probe cycles are with respect to the mean tolerance in **Q98x** and **Q99x**. The values are thus the same as the correction values executed by the cycle if the input parameters **Q1120** and **Q1121** are programmed. If no automatic evaluation is active, the control will save the values with respect to the mean tolerance in the programmed Q parameters. You can further process these values.

Example



11 TCH PROBE 1411PROBING TWO CIRCLES ~	Define the cycle
Q1100=+30 ;1ST POINT REF AXIS ~	Nominal position 1 in main axis
Q1101=+50 ;1ST POINT MINOR AXIS ~	Nominal position 1 in secondary axis
Q1102=-5 ;1ST POINT TOOL AXIS ~	Nominal position 1 in tool axis
QS1116="+8-2-1" ;DIAMETER 1 ~	Nominal dimension 1 including tolerance
Q1103=+75 ;2ND POINT REF AXIS ~	Nominal position 2 in main axis
Q1104=+50 ;2ND POINT MINOR AXIS ~	Nominal position 2 in secondary axis
QS1105=-5 ;2ND POINT TOOL AXIS ~	Nominal position 2 in tool axis
QS1117="+8-2-1" ;DIAMETER 2 ~	Nominal dimension 2 including tolerance
Q1115=+0 ;GEOMETRY TYPE ~	
Q423=+4 ;NO. OF PROBE POINTS ~	
Q325=+0 ;STARTING ANGLE ~	
Q1119=+360 ;ANGULAR LENGTH ~	
Q320=+2 ;SET-UP CLEARANCE ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q1125=+2 ;CLEAR. HEIGHT MODE ~	
Q309=2 ;ERROR REACTION ~	
Q1126=+0 ;ALIGN ROTARY AXIS ~	
Q1120=+0 ;TRANSER POSITION ~	
Q1121=+0 ;CONFIRM ROTATION	

Transferring the actual position

You can determine the actual position in advance and define it as the actual position for the touch probe cycle. Then, both the nominal position and the actual position will be transferred. Based on the difference, the cycle calculates the required compensation values and applies tolerance monitoring.

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



Programming and operating notes:

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

Some of these nominal positions are defined with tolerance monitoring and actual position.

5 TCH PROBE 1410 PROBING ON EDGE ~	
QS1100="10+0.02@10.0123"	;1ST POINT REF AXIS ~
QS1101="50@50.0321"	;1ST POINT MINOR AXIS ~
QS1102="-10-0.2+0.2@Q1900"	;1ST POINT TOOL AXIS ~
QS1103="30+0.02@30.0134"	;2ND POINT REF AXIS ~
QS1104="50@50.534"	;2ND POINT MINOR AXIS ~
QS1105="-10-0.02@Q1901"	;2ND POINT TOOL AXIS ~
Q372=+2	;PROBING DIRECTION ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

4.3 Cycle 1420 PROBING IN PLANE

ISO programming

G1420

Application

Touch probe cycle **1420** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 309

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

Further information: "Semi-automatic mode", Page 55

- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

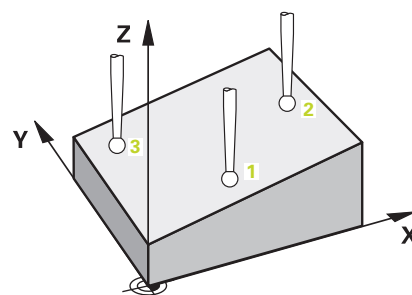
Further information: "Evaluation of tolerances", Page 60

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

Further information: "Transferring the actual position", Page 63

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 It then moves in the working plane to touch point **2** to measure the actual value of the second touch point in the plane.
- 5 The touch probe returns to the clearance height (depending on **Q1125**), then moves in the working plane to touch point **3** and measures the actual position of the third point of the plane.
- 6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:



Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q956 to Q958	Measured position 3 in the main axis, secondary axis, and tool axis
Q961 to Q963	Measured spatial angle SPA, SPB, and SPC in the W-CS
Q980 to Q982	Measured deviation from the first touch point
Q983 to Q985	Measured deviation from the second touch point
Q986 to Q988	Third measured deviation of the positions
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the first touch point</p>
Q971	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the second touch point</p>
Q972	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the third touch point</p>

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- ▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three touch points are not positioned on a straight line.
- The nominal spatial angle results from the defined nominal positions. The cycle saves the measured spatial angle in parameters **Q961** to **Q963**. For the transfer to the 3D basic rotaton, the control uses the difference between the measured spatial angle and the nominal spatial angle.



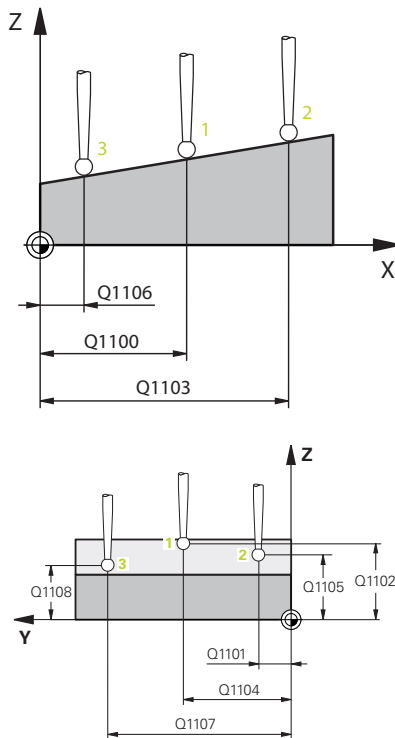
- HEIDENHAIN recommends avoiding the use of axis angles in this cycle!

Aligning the rotary table axes:

- Alignment of rotary axes is only possible if two rotary axes are available in the kinematics.
- To align the rotary axes (**Q1126** unequal to 0), the rotation must be accepted (**Q1121** unequal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the first touch point in the main axis of the working plane

Input: **-99999.9999...+99999.9999** or **?, -, +** or **@**

- **?**: Semi-automatic mode, see Page 55
- **-, +**: Evaluation of the tolerance, see Page 60
- **@**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the first touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1103 2nd noml. position of ref axis?

Absolute nominal position of the second touch point in the main axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1104 2nd noml. position of minor axis?

Absolute nominal position of the second touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1105 2nd nominal pos. of tool axis?

Absolute nominal position of the second touch point in the tool axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1106 3rd noml. position of ref axis?

Absolute nominal position of the third touch point in the main axis of the working plane.

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Help graphic

Parameter

Q1107 3rd noml. position minor axis?

Absolute nominal position of the third touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1108 3rd nominal position tool axis?

Absolute nominal position of the third touch point in the tool axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q372 Probe direction (-3 to +3)?

Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction.

Input: **-3, -2, -1, +1, +2, +3**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

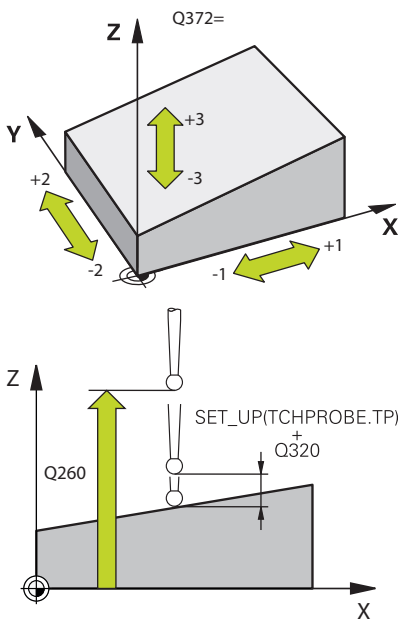
-1: Do not move to the clearance height.

0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**

Input: **-1, 0, +1, +2**



Help graphic**Parameter****Q309 Reaction to tolerance error?**

Reaction when tolerance is exceeded:

- 0:** Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.
- 1:** Interrupt program run when tolerance is exceeded. The control opens a window with the results.
- 2:** The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1126 Align rotary axes?

Position the rotary axes for inclined machining:

- 0:** Retain the current position of the rotary axis.
- 1:** Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.
- 2:** Automatically position the rotary axis without orienting the tool tip (**TURN**).

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

- 0:** No correction
- 1:** Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.
- 2:** Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.
- 3:** Correction based on 3rd touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 3rd touch point.
- 4:** Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

Input: **0, 1, 2, 3, 4**

Q1121 Confirm basic rotation?

Define whether the control will use the determined misalignment as a basic rotation:

- 0:** No basic rotation
- 1:** Set basic rotation: The control will save the basic rotation

Input: **0, 1**

Example

11 TCH PROBE 1420 PROBING IN PLANE ~	
Q1100=+0	;1ST POINT REF AXIS ~
Q1101=+0	;1ST POINT MINOR AXIS ~
Q1102=+0	;1ST POINT TOOL AXIS ~
Q1103=+0	;2ND POINT REF AXIS ~
Q1104=+0	;2ND POINT MINOR AXIS ~
Q1105=+0	;2ND POINT TOOL AXIS ~
Q1106=+0	;3RD POINT REF AXIS ~
Q1107=+0	;3RD POINT MINOR AXIS ~
Q1108=+0	;3RD POINT 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

4.4 Cycle 1410 PROBING ON EDGE

ISO programming

G1410

Application

Touch probe cycle **1410** allows you to determine workpiece misalignment by probing two points on an edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 309

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

Further information: "Semi-automatic mode", Page 55

- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

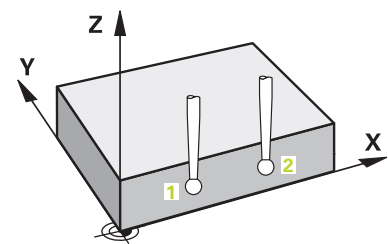
Further information: "Evaluation of tolerances", Page 60

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

Further information: "Transferring the actual position", Page 63

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 The control offsets the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.
- 4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 5 The touch probe then moves to the next touch point **2** and probes again.
- 6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:



Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviation from the first touch point
Q983 to Q985	Measured deviation from the second touch point
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the first touch point</p>
Q971	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the second touch point</p>

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- ▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Note about rotary axes:

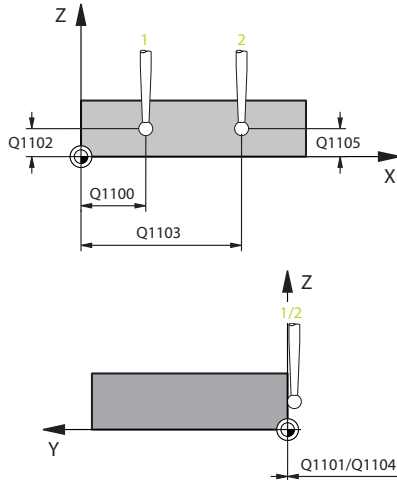
- If you determine the basic rotation in a tilted machining plane, then note the following:
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (no. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the first touch point in the main axis of the working plane

Input: **-99999.9999...+99999.9999** or **?, -, +** or **@**

- **?**: Semi-automatic mode, see Page 55
- **-, +**: Evaluation of the tolerance, see Page 60
- **@**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the first touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1103 2nd noml. position of ref axis?

Absolute nominal position of the second touch point in the main axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1104 2nd noml. position of minor axis?

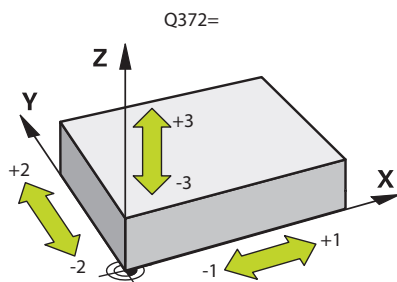
Absolute nominal position of the second touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1105 2nd nominal pos. of tool axis?

Absolute nominal position of the second touch point in the tool axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

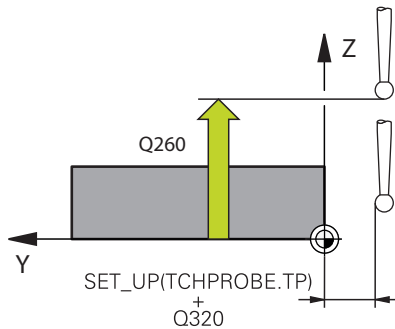


Q372 Probe direction (-3 to +3)?

Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction.

Input: **-3, -2, -1, +1, +2, +3**

Help graphic



Parameter

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Help graphic**Parameter****Q1126 Align rotary axes?**

Position the rotary axes for inclined machining:

0: Retain the current position of the rotary axis.

1: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.

2: Automatically position the rotary axis without orienting the tool tip (**TURN**).

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

2: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

3: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

Input: **0, 1, 2, 3**

Q1121 CONFIRM ROTATION?

Define whether the control should use the determined misalignment:

0: No basic rotation

1: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation.

2: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset.

Input: **0, 1, 2**

Example

11 TCH PROBE 1410 PROBING ON EDGE ~	
Q1100=+0	;1ST POINT REF AXIS ~
Q1101=+0	;1ST POINT MINOR AXIS ~
Q1102=+0	;1ST POINT TOOL AXIS ~
Q1103=+0	;2ND POINT REF AXIS ~
Q1104=+0	;2ND POINT MINOR AXIS ~
Q1105=+0	;2ND POINT TOOL AXIS ~
Q372=+1	;PROBING DIRECTION ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

4.5 Cycle 1411 PROBING TWO CIRCLES

ISO programming

G1411

Application

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

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 309

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

Further information: "Semi-automatic mode", Page 55

- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

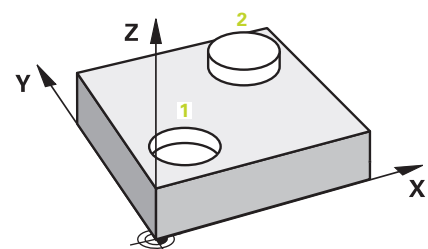
Further information: "Evaluation of tolerances", Page 60

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

Further information: "Transferring the actual position", Page 63

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch object **1** at **FMAX** (from the touch probe table), using positioning logic.
Further information: "Positioning logic", Page 45
- 2 With **FMAX** (from the touch probe table), the touch probe moves to the entered measuring height **Q1102**.
- 3 Depending on the number of probing processes **Q423**, the touch probe acquires the touch points and ascertains the first hole center or stud center.
- 4 If you have programmed the **CLEAR. HEIGHT MODE Q1125**, the control will move the touch probe to the clearance height between the touch points and at the end of the probing object. During this process, the control positions the touch probe at **FMAX** from the touch probe table.
- 5 The control positions the touch probe to the pre-position of the second probing object **2** and repeats steps 2 to 4.
- 6 After that, the control saves the measured values in the following Q parameters:



Q parameter number	Meaning
Q950 to Q952	Measured circle center point 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured circle center point 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q966 to Q967	Measured first and second diameters
Q980 to Q982	Measured deviation of the first circle center
Q983 to Q985	Measured deviation of the second center
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q996 to Q997	Measured deviation of the diameters
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the first circle center</p>
Q971	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the second circle center</p>
Q973	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from Diameter 1</p>
Q974	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from Diameter 2</p>



Operating note:

- If the hole is too small to achieve the programmed set-up clearance, a window opens. In the window, the control displays the nominal dimension of the hole, the calibrated ball-tip radius, and the achievable set-up clearance.

You have the following options:

- If there is no danger of collision, press **NC Start** to execute the cycle with the values from the dialog. The active set-up clearance is reduced to the displayed value only for this object.
- You can cancel the cycle by pressing Cancel.

Notes

NOTICE**Danger of collision!**

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- ▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE**Danger of collision!**

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

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

Note about rotary axes:

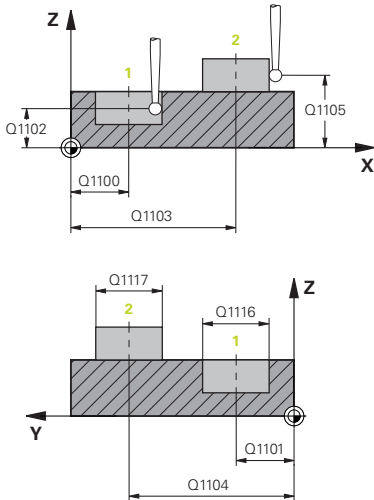
- If you determine the basic rotation in a tilted machining plane, then note the following:
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (no. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the center in the main axis of the working plane.

Input: **-99999.9999...+99999.9999** or enter **?, +, -** or **@**:

- **"?..."**: Semi-automatic mode, see Page 55
- **"...-...+..."**: Evaluation of the tolerance, see Page 60
- **"...@..."**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the center in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1116 Diameter of 1st position?

Diameter of the first hole or the first stud

Input: **0...9999.9999** or optional input:

- **"...-...+..."**: Evaluation of the tolerance, see Page 60

Q1103 2nd noml. position of ref axis?

Absolute nominal position of the center in the main axis of the working plane.

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1104 2nd noml. position of minor axis?

Absolute nominal position of the center in the secondary axis of the working plane.

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1105 2nd nominal pos. of tool axis?

Absolute nominal position of the second touch point in the tool axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Help graphic

Parameter

Q1117 Diameter of 2nd position?

Diameter of the second hole or the second stud

Input: **0...9999.9999** or optional input:

"...-...+...": Evaluation of the tolerance, see Page 60

Q1115 Geometry type (0-3)?

Type of object to be probed:

0: Position 1 = hole, and position 2 = hole

1: Position 1 = stud, and position 2 = stud

2: Position 1 = hole, and position 2 = stud

3: Position 1 = stud, and position 2 = hole

Input: **0, 1, 2, 3**

Q423 Number of probes?

Number of touch points on the diameter

Input: **3, 4, 5, 6, 7, 8**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q1119 Arc angular length?

Angular range in which the touch points are distributed.

Input: **-359.999...+360.000**

Q320 Set-up clearance?

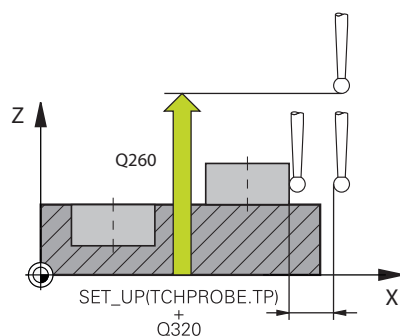
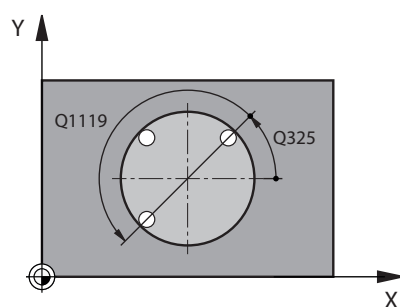
Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic**Parameter**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1126 Align rotary axes?

Position the rotary axes for inclined machining:

0: Retain the current position of the rotary axis.

1: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.

2: Automatically position the rotary axis without orienting the tool tip (**TURN**).

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

2: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

3: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

Input: **0, 1, 2, 3**

Help graphic

Parameter

Q1121 CONFIRM ROTATION?

Define whether the control should use the determined misalignment:

0: No basic rotation

1: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation.

2: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset.

Input: **0, 1, 2**

Example

11 TCH PROBE 1411 PROBING TWO CIRCLES ~	
Q1100=+0	;1ST POINT REF AXIS ~
Q1101=+0	;1ST POINT MINOR AXIS ~
Q1102=+0	;1ST POINT TOOL AXIS ~
Q1116=+0	;DIAMETER 1 ~
Q1103=+0	;2ND POINT REF AXIS ~
Q1104=+0	;2ND POINT MINOR AXIS ~
Q1105=+0	;2ND POINT TOOL AXIS ~
Q1117=+0	;DIAMETER 2 ~
Q1115=+0	;GEOMETRY TYPE ~
Q423=+4	;NO. OF PROBE POINTS ~
Q325=+0	;STARTING ANGLE ~
Q1119=+360	;ANGULAR LENGTH ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

4.6 Cycle 1412 INCLINED EDGE PROBING

ISO programming

G1412

Application

Touch probe cycle **1412** allows you to determine workpiece misalignment by probing two points on an inclined edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 309

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

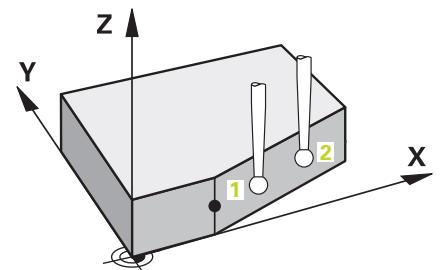
Further information: "Semi-automatic mode", Page 55

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

Further information: "Transferring the actual position", Page 63

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 The control retracts the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.
- 4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 5 The touch probe then moves to the touch point **2** and probes again.
- 6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:



Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviation from the first touch point
Q983 to Q985	Measured deviation from the second touch point
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the first touch point</p>
Q971	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the second touch point</p>

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- ▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program a tolerance in **Q1100**, **Q1101**, or **Q1102**, then this tolerance applies to the programmed nominal positions instead of to the touch points along the inclined edge. Use the **TOLERANCE QS400** parameter to program a tolerance for the surface normal along the inclined edge.

Note about rotary axes:

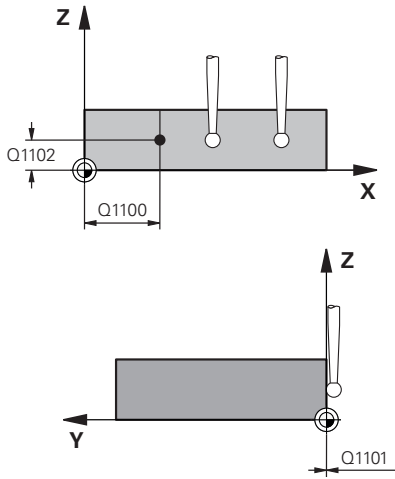
- When you determine the basic rotation in a tilted working plane, keep the following in mind:
 - If the current coordinates of the rotary axes and the defined tilt angles (3D ROT menu) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
 - If the current coordinates of the rotary axes and the defined tilt angles (3D ROT menu) do not match, the working plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** in dependence on the tool axis.
- In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines whether the control checks the matching of the tilting situation. If no check is configured, the control always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position at which the inclined edge begins in the main axis.

Input: **-99999.9999...+99999.9999** or **?, +, -** or **@**

- **?**: Semi-automatic mode, see Page 55
- **-, +**: Evaluation of the tolerance, see Page 60
- **@**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position at which the inclined edge begins in the secondary axis.

Input: **-99999.9999...+99999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

QS400 Tolerance value?

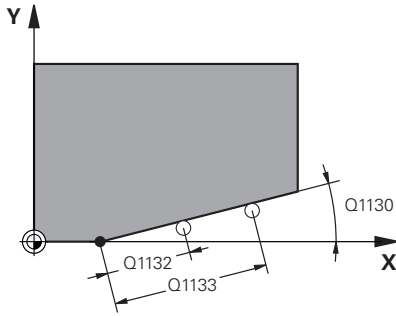
Tolerance band monitored by the cycle. The tolerance defines the deviation permitted for the surface normals along the inclined edge. The control determines this deviation using the nominal coordinate and the actual coordinate of the workpiece.

Examples:

- **QS400 = "0.4-0.1"**: Upper dimension = Nominal coordinate +0.4; Lower dimension = Nominal coordinate -0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate -0.1"
- **QS400 = " "**: No monitoring of the tolerance.
- **QS400 = "0"**: No monitoring of the tolerance.
- **QS400 = "0.1+0.1"**: No monitoring of the tolerance.

Input: Max. **255** characters

Help graphic



Parameter

Q1130 Nominal angle for 1st line?

Nominal angle of the first straight line

Input: **-180...+180**

Q1131 Probing direction for 1st line?

Probing direction for the first edge:

+1: Rotates the probing direction by $+90^\circ$ to the nominal angle **Q1130** and probes at right angles to the nominal edge.

-1: Rotates the probing direction by -90° to the nominal angle **Q1130** and probes at right angles to the nominal edge.

Input: **-1, +1**

Q1132 First distance on 1st line?

Distance between the beginning of the inclined edge and the first touch point. This value has an incremental effect.

Input: **-999.999...+999.999**

Q1133 Second distance on 1st line?

Distance between the beginning of the inclined edge and the second touch point. This value has an incremental effect.

Input: **-999.999...+999.999**

Q1139 Plane for object (1-3)?

Plane in which the control interprets the nominal angle **Q1130** and the probing direction **Q1131**.

1: YZ plane

2: ZX plane

3: XY plane

Input: **1, 2, 3**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

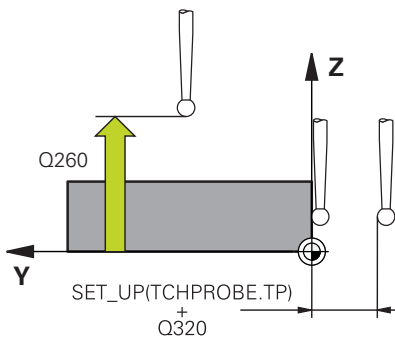
-1: Do not move to the clearance height.

0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**

Input: **-1, 0, +1, +2**



Help graphic

Parameter

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1126 Align rotary axes?

Position the rotary axes for inclined machining:

0: Retain the current position of the rotary axis.

1: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.

1: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

2: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

3: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

Input: **0, 1, 2, 3**

Help graphic

Parameter

Q1121 CONFIRM ROTATION?

Define whether the control should use the determined misalignment:

0: No basic rotation

1: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation.

2: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset.

Input: **0, 1, 2**

Example

11 TCH PROBE 1412 INCLINED EDGE PROBING ~	
Q1100=+20	;1ST POINT REF AXIS ~
Q1101=+0	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS400="+0.1-0.1"	;TOLERANCE ~
Q1130=+30	;NOMINAL ANGLE, 1ST LINE ~
Q1131=+1	;PROBE DIRECTION, 1ST LINE ~
Q1132=+10	;FIRST DISTANCE, 1ST LINE ~
Q1133=+20	;SECOND DISTANCE, 1ST LINE ~
Q1139=+3	;OBJECT PLANE ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

4.7 Cycle 1416 INTERSECTION PROBING

ISO programming

G1416

Application

Touch probe cycle **1416** allows you to determine the intersection of two edges. You can execute the cycle in all three machining planes XY, XZ and YZ. The cycle requires a total of four touch points and two positions per edge. You can select the sequence of the edges as desired.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 309

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

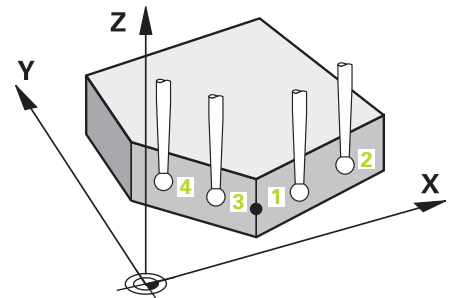
Further information: "Semi-automatic mode", Page 55

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

Further information: "Transferring the actual position", Page 63

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control positions the touch probe to the next touch point.
- 5 The control positions the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 6 The control repeats Steps 3 to 5 until all four touch points are measured.
- 7 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.



Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis and tool axis
Q956 to Q958	Measured position 3 in the main axis, secondary axis and tool axis
Q959 to Q960	Measured intersection in the main axis and secondary axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviation of the first touch point in the main axis, auxiliary axis and tool axis
Q983 to Q985	Measured deviation of the second touch point in the main axis, auxiliary axis and tool axis
Q986 to Q988	Measured deviation of the third touch point in the main axis, auxiliary axis and tool axis
Q989 to Q990	Measured deviations of the intersection in the main axis and secondary axis
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation from the 1st touch point</p>
Q971	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation from the 2nd touch point</p>
Q972	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation from the 3rd touch point</p>

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

- ▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Note about rotary axes:

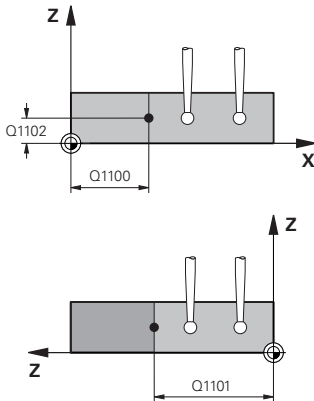
- If you determine the basic rotation in a tilted machining plane, then note the following:
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (no. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position in the main axis at which the two edges intersect.

Input: **-99999.9999...+99999.9999** or ? or @

- ? : Semi-automatic mode, see Page 55
- @ : Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position in the secondary axis at which the two edges intersect.

Input: **-99999.9999...+99999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the touch points in the tool axis

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

QS400 Tolerance value?

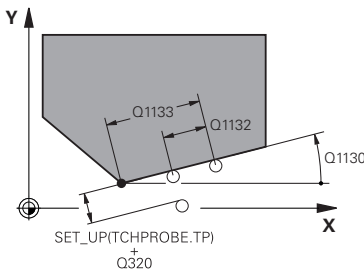
Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normal along the first edge. The control determines the deviation using the nominal coordinates and the actual coordinates of the part.

Examples:

- **QS400 = "0.4-0.1"**: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate -0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate -0.1"
- **QS400 = " "**: No monitoring of the tolerance.
- **QS400 = "0"**: No monitoring of the tolerance.
- **QS400 = "0.1+0.1"**: No monitoring of the tolerance.

Input: Max. **255** characters

Help graphic



Parameter

Q1130 Nominal angle for 1st line?

Nominal angle of the first straight line

Input: **-180...+180**

Q1131 Probing direction for 1st line?

Probing direction for the first edge:

+1: Rotates the probing direction by $+90^\circ$ to the nominal angle **Q1130** and probes at right angles to the nominal edge.

-1: Rotates the probing direction by -90° to the nominal angle **Q1130** and probes at right angles to the nominal edge.

Input: **-1, +1**

Q1132 First distance on 1st line?

Distance between the intersection and the first touch point on the first edge. This value has an incremental effect.

Input: **-999.999...+999.999**

Q1133 Second distance on 1st line?

Distance between the intersection and the second touch point on the first edge. This value has an incremental effect.

Input: **-999.999...+999.999**

QS401 Tolerance value 2?

Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normals along the second edge. The control determines this deviation using the nominal coordinate and the actual coordinate of the workpiece.

Input: Max. **255** characters

Q1134 Nominal angle for 2nd line?

Nominal angle of the first straight line

Input: **-180...+180**

Q1135 Probing direction for 2nd line?

Probing direction for the second edge:

+1: Rotates the probing direction by $+90^\circ$ relative to the nominal angle **Q1134** and probes at right angles relative to the nominal edge.

-1: Rotates the probing direction by -90° relative to the nominal angle **Q1134**, and probes at right angles relative to the nominal edge.

Input: **-1, +1**

Q1136 First distance on 2nd line?

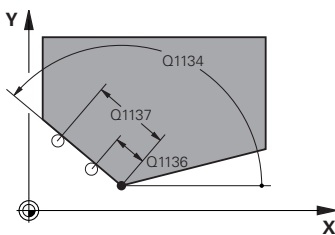
Distance between the intersection and the first touch point on the second edge. This value has an incremental effect.

Input: **-999.999...+999.999**

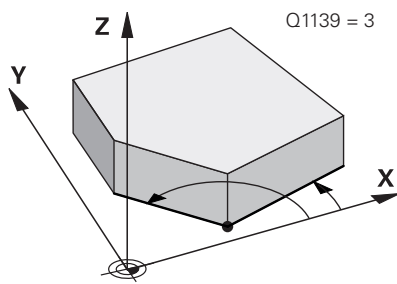
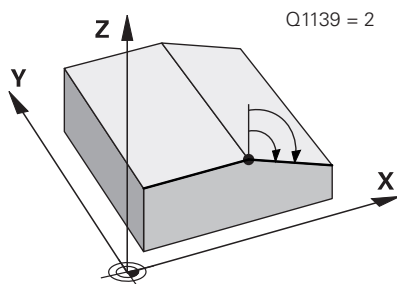
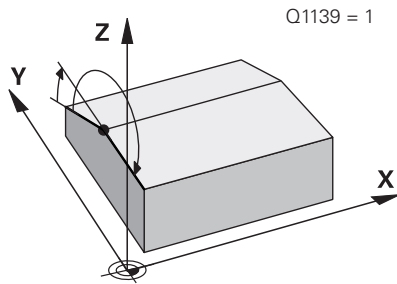
Q1137 Second distance on 2nd line?

Distance between the intersection and the second touch point on the second edge. This value has an incremental effect.

Input: **-999.999...+999.999**



Help graphic



Parameter

Q1139 Plane for object (1-3)?

Plane in which the control interprets the nominal angle **Q1130** and **Q1134**, as well as the probing direction **Q1131** and **Q1135**.

1: YZ plane

2: ZX plane

3: XY plane

Input: **1, 2, 3**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Help graphic

Parameter

Q1126 Align rotary axes?

Position the rotary axes for inclined machining:

0: Retain the current position of the rotary axis.

1: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.

2: Automatically position the rotary axis without orienting the tool tip (**TURN**).

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction of the active preset based on the point of intersection. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the intersection.

Input: **0, 1**

Q1121 CONFIRM ROTATION?

Define whether the control should use the determined misalignment:

0: No basic rotation

1: Set the basic rotation: The control transfers the misalignment of the first edge to the preset table as a basic transformation.

2: Execute rotary table rotation: The control transfers the misalignment of the first edge to the preset table as an offset.

3: Set the basic rotation: The control transfers the misalignment of the second edge to the preset table as a basic transformation.

4: Execute rotary table rotation: The control transfers the misalignment of the second edge to the preset table as an offset.

5: Set basic rotation: The control transfers the misalignment from the mean deviations of both edges to the preset table as a basic transformation.

6: Execute rotary table rotation: The control transfers the misalignment from the mean deviations of both edges to the preset table as an offset.

Input: **0, 1, 2, 3, 4, 5, 6**

Example

11 TCH PROBE 1416 INTERSECTION PROBING ~	
Q1100=+50	;1ST POINT REF AXIS ~
Q1101=+10	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS400="0"	;TOLERANCE ~
Q1130=+45	;NOMINAL ANGLE, 1ST LINE ~
Q1131=+1	;PROBE DIRECTION, 1ST LINE ~
Q1132=+10	;FIRST DISTANCE, 1ST LINE ~
Q1133=+25	;SECOND DISTANCE, 1ST LINE ~
QS401="0"	;TOLERANCE 2 ~
Q1134=+135	;NOMINAL ANGLE, 2ND LINE ~
Q1135=-1	;PROBE DIRECTION, 2ND LINE ~
Q1136=+10	;FIRST DISTANCE, 2ND LINE ~
Q1137=+25	;SECOND DISTANCE, 2ND LINE ~
Q1139=+3	;OBJECT PLANE ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

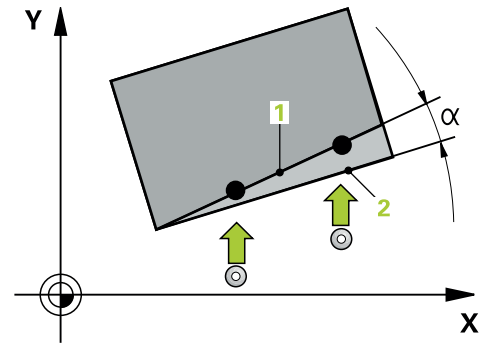
4.8 Fundamentals of touch probe cycles 400 to 405

Characteristics common to all touch probe cycles for measuring workpiece misalignment

In Cycles **400**, **401**, and **402**, you can use parameter **Q307 Preset value for rotation angle** to define whether the measurement result will be corrected by a known angle α (see figure). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.



These cycles do not work with 3D ROT! In such a case, use cycles **14xx**. **Further information:** "Touch probe cycles 14xx: fundamentals", Page 53



4.9 Cycle 400 BASIC ROTATION

ISO programming

G400

Application

Touch probe cycle **400** determines a workpiece misalignment by measuring two points, which must lie on a straight line. With the basic rotation function, the control compensates the measured value.



Instead of Cycle **400 BASIC ROTATION**, HEIDENHAIN recommends using the more powerful cycles below:

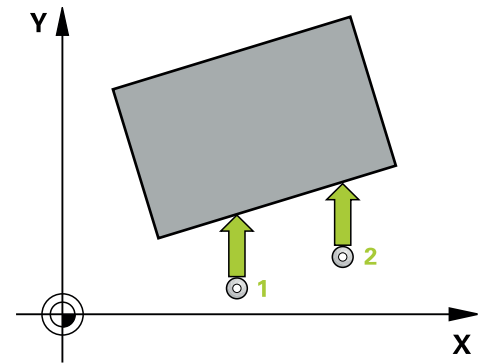
- **1410 PROBING ON EDGE**
- **1412 INCLINED EDGE PROBING**

Related topics

- Cycle **1410 PROBING ON EDGE**
Further information: "Cycle 1410 PROBING ON EDGE", Page 71
- Cycle **1412 INCLINED EDGE PROBING**
Further information: "Cycle 1412 INCLINED EDGE PROBING", Page 87

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and performs the basic rotation it determined.



Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

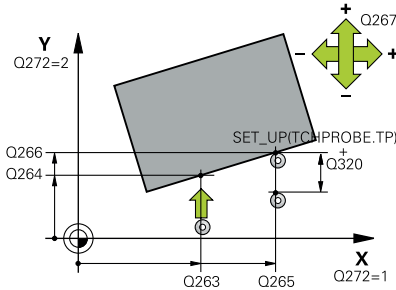
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis

Input: **1, 2**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

- 1: Negative traverse direction
- +1: Positive traverse direction

Input: **-1, +1**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

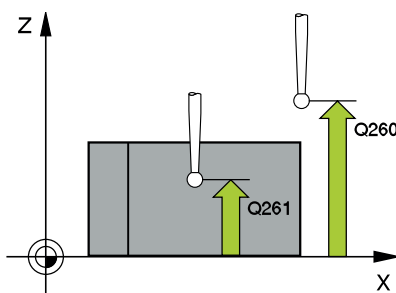
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic

Parameter

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q307 Preset value for rotation angle

If the misalignment is measured relative to any straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. This value has an absolute effect.

Input: **-360.000...+360.000**

Q305 Preset number in table?

Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter **Q305 = 0**, the control automatically stores the calculated basic rotation in the ROT menu of the Manual Operation mode.

Input: **0...99999**

Example

11 TCH PROBE 400 BASIC ROTATION ~	
Q263=+10	;1ST POINT 1ST AXIS ~
Q264=+3.5	;1ST POINT 2ND AXIS ~
Q265=+25	;2ND POINT 1ST AXIS ~
Q266=+2	;2ND PNT IN 2ND AXIS ~
Q272=+2	;MEASURING AXIS ~
Q267=+1	;TRAVERSE DIRECTION ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q307=+0	;PRESET ROTATION ANG. ~
Q305=+0	;NUMBER IN TABLE

4.10 Cycle 401 ROT OF 2 HOLES

ISO programming

G401

Application

Touch probe cycle **401** measures the center points of two holes. The control then calculates the angle between the main axis of the working plane and the line connecting the hole center points. With the basic rotation function, the control compensates for the calculated value. As an alternative, you can also compensate for the determined misalignment by rotating the rotary table.

i Instead of Cycle **401 ROT OF 2 HOLES**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

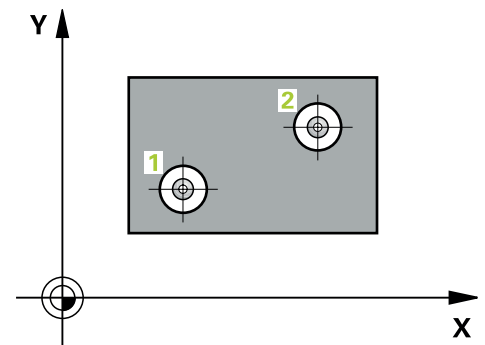
Related topics

- Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES", Page 79

Cycle run

- 1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic
Further information: "Positioning logic", Page 45
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.



Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

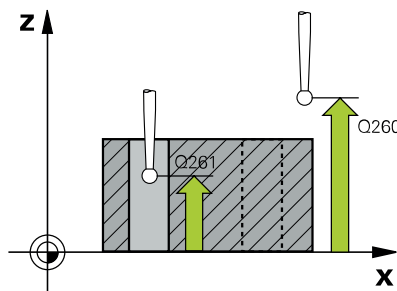
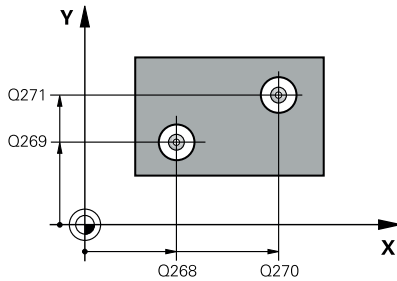
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
 - C for tool axis Z
 - B for tool axis Y
 - A for tool axis X

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q268 1st hole: center in 1st axis?

Center of the first hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q269 1st hole: center in 2nd axis?

Center of the first hole in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q270 2nd hole: center in 1st axis?

Center of the second hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q271 2nd hole: center in 2nd axis?

Center of the second hole in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q307 Preset value for rotation angle

If the misalignment is measured relative to any straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. This value has an absolute effect.

Input: **-360.000...+360.000**

Help graphic

Parameter

Q305 Number in table?

Enter the number of a row in the preset table. The control will make the corresponding entry in the following row:

Q305 = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.

Q305 > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**).

Q305 depends on the following parameters:

- **Q337 = 0** and, at the same time, **Q402 = 0:** A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, the basic rotation is entered in the **SPC** column).
- **Q337 = 0** and, at the same time, **Q402 = 1:** The parameter **Q305** is not effective.
- **Q337 = 1:** The parameter **Q305** has the effect described above.

Input: **0...99999**

Q402 Basic rotation/alignment (0/1)

Define whether the control will set the determined misalignment as a basic rotation or will compensate it by rotating the rotary table:

0: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**)

1: Rotate the rotary table: An entry will be made in the corresponding **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column); in addition, the corresponding axis will be rotated

Input: **0, 1**

Q337 Set to zero after alignment?

Define whether the control will set the position display of the corresponding rotary axis to 0 after the alignment:

0: The position display is not set to 0 after the alignment

1: After the alignment, the position display is set to 0, provided you have defined **Q402 = 1**

Input: **0, 1**

Example

11 TCH PROBE 401 ROT OF 2 HOLES ~	
Q268=-37	;1ST CENTER 1ST AXIS ~
Q269=+12	;1ST CENTER 2ND AXIS ~
Q270=+75	;2ND CENTER 1ST AXIS ~
Q271=+20	;2ND CENTER 2ND AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+20	;CLEARANCE HEIGHT ~
Q307=+0	;PRESET ROTATION ANG. ~
Q305=+0	;NUMBER IN TABLE ~
Q402=+0	;COMPENSATION ~
Q337=+0	;SET TO ZERO

4.11 Cycle 402 ROT OF 2 STUDS

ISO programming

G402

Application

Touch probe cycle **402** measures the center points of two cylindrical studs. The control then calculates the angle between the main axis of the working plane and the line connecting the stud center points. With the basic rotation function, the control compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.



Instead of Cycle **402 ROT OF 2 STUDS**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

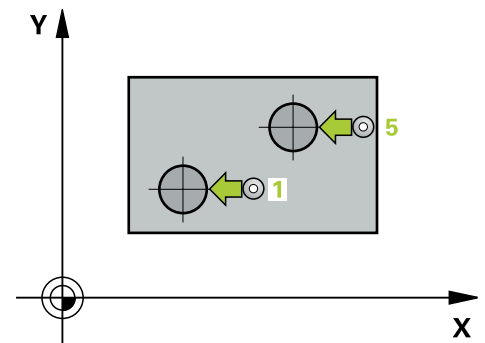
Related topics

- Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES", Page 79

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Then the touch probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves along a circular arc between the touch points, each of which is offset by 90°.
- 3 The touch probe returns to the clearance height and then moves to the touch point **5** of the second stud.
- 4 The control moves the touch probe to the entered **measuring height 2** and probes four points to determine the center of the second stud.
- 5 Then the control returns the touch probe to the clearance height and performs the calculated basic rotation.



Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

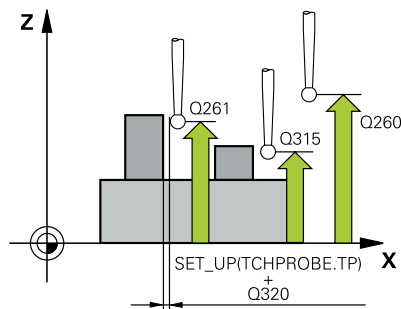
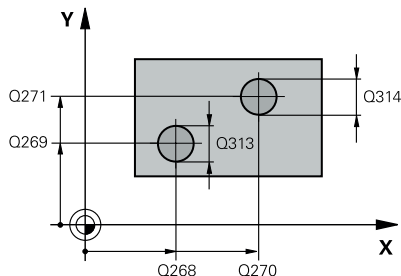
- ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
 - ▶ Reset any coordinate transformations beforehand.
-
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
 - The control will reset an active basic rotation at the beginning of the cycle.
 - If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
 - C for tool axis Z
 - B for tool axis Y
 - A for tool axis X

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q268 1st stud: center in 1st axis?

Center of the first stud in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q269 1st stud: center in 2nd axis?

Center of the first stud in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q313 Diameter of stud 1?

Approximate diameter of the first stud. Enter a value that is more likely to be too large than too small.

Input: **0...99999.9999**

Q261 Meas. height stud 1 in TS axis?

Coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 1 will be measured. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q270 2nd stud: center in 1st axis?

Center of the second stud in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q271 2nd stud: center in 2nd axis?

Center of the second stud in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q314 Diameter of stud 2?

Approximate diameter of the second stud. Enter a value that is more likely to be too large than too small.

Input: **0...99999.9999**

Q315 Meas. height stud 2 in TS axis?

Coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 2 will be measured. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic**Parameter****Q301 Move to clearance height (0/1)?**

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q307 Preset value for rotation angle

If the misalignment is measured relative to any straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. This value has an absolute effect.

Input: **-360.000...+360.000**

Q305 Number in table?

Enter the number of a row in the preset table. The control will make the corresponding entry in the following row:

Q305 = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.

Q305 > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**).

Q305 depends on the following parameters:

- **Q337 = 0** and, at the same time, **Q402 = 0:** A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, the basic rotation is entered in the **SPC** column).
- **Q337 = 0** and, at the same time, **Q402 = 1:** The parameter **Q305** is not effective.
- **Q337 = 1:** The parameter **Q305** has the effect described above.

Input: **0...99999**

Help graphic

Parameter

Q402 Basic rotation/alignment (0/1)

Define whether the control will set the determined misalignment as a basic rotation or will compensate it by rotating the rotary table:

0: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**)

1: Rotate the rotary table: An entry will be made in the corresponding **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column); in addition, the corresponding axis will be rotated

Input: **0, 1**

Q337 Set to zero after alignment?

Define whether the control will set the position display of the corresponding rotary axis to 0 after the alignment:

0: The position display is not set to 0 after the alignment

1: After the alignment, the position display is set to 0, provided you have defined **Q402 = 1**

Input: **0, 1**

Example

11 TCH PROBE 402 ROT OF 2 STUDS ~	
Q268=-37	;1ST CENTER 1ST AXIS ~
Q269=+12	;1ST CENTER 2ND AXIS ~
Q313=+60	;DIAMETER OF STUD 1 ~
Q261=-5	;MEAS. HEIGHT STUD 1 ~
Q270=+75	;2ND CENTER 1ST AXIS ~
Q271=+20	;2ND CENTER 2ND AXIS ~
Q314=+60	;DIAMETER OF STUD 2 ~
Q315=-5	;MEAS. HEIGHT STUD 2 ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q307=+0	;PRESET ROTATION ANG. ~
Q305=+0	;NUMBER IN TABLE ~
Q402=+0	;COMPENSATION ~
Q337=+0	;SET TO ZERO

4.12 Cycle 403 ROT IN ROTARY AXIS

ISO programming

G403

Application

Touch probe cycle **403** determines a workpiece misalignment by measuring two points, which must lie on a straight line. The control compensates for the determined misalignment by rotating the A, B, or C axis. The workpiece can be clamped in any position on the rotary table.



Instead of Cycle **403 ROT IN ROTARY AXIS**, HEIDENHAIN recommends using the more powerful cycles below:

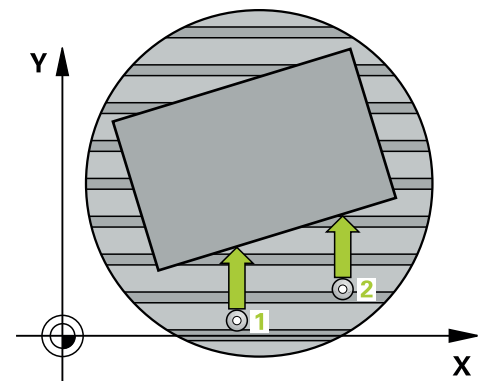
- **1410 PROBING ON EDGE**
- **1412 INCLINED EDGE PROBING**

Related topics

- Cycle **1410 PROBING ON EDGE**
Further information: "Cycle 1410 PROBING ON EDGE", Page 71
- Cycle **1412 INCLINED EDGE PROBING**
Further information: "Cycle 1412 INCLINED EDGE PROBING", Page 87

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and rotates the rotary axis, which was defined in the cycle, by the measured value. Optionally, you can specify whether the control is to set the determined rotation angle to 0 in the preset table or in the datum table.



Notes

NOTICE**Danger of collision!**

If the control positions the rotary axis automatically, a collision might occur.

- ▶ Check for possible collisions between the tool and any elements positioned on the table
- ▶ Select the clearance height to prevent collisions

NOTICE**Danger of collision!**

If you set parameter **Q312** Axis for compensating movement? to 0, then the cycle will automatically determine the rotary axis to be aligned (recommended setting). When doing so, it determines an angle that depends on the sequence of the touch points. The measured angle goes from the first to the second touch point. If you select the A, B or C axis as compensation axis in parameter **Q312**, the cycle determines the angle, regardless of the sequence of the touch points. The calculated angle is in the range of -90° to $+90^\circ$. There is a risk of collision!

- ▶ After alignment, check the position of the rotary axis.

NOTICE**Danger of collision!**

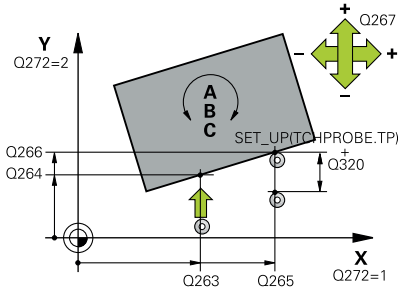
When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis
- 3: Touch probe axis = measuring axis

Input: **1, 2, 3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

- 1: Negative traverse direction
- +1: Positive traverse direction

Input: **-1, +1**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

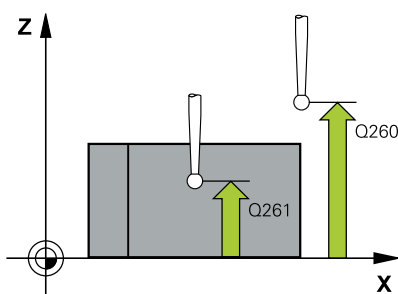
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic

Parameter

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q312 Axis for compensating movement?

Define the rotary axis in which the control will compensate the measured misalignment:

0: Automatic mode – the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting!

4: Compensate misalignment with rotary axis A

5: Compensate misalignment with rotary axis B

6: Compensate misalignment with rotary axis C

Input: **0, 4, 5, 6**

Q337 Set to zero after alignment?

Define whether the control will set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment.

0: Do not set the angle of the rotary axis to 0 in the table after the alignment

1: Set the angle of the rotary axis to 0 in the table after the alignment

Input: **0, 1**

Q305 Number in table?

Specify the number of the row in the preset table in which the control will enter the basic rotation.

Q305 = 0: The rotary axis is zeroed in row number 0 of the preset table. The control will make an entry in the **OFFSET** column. In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.

Q305 > 0: Specify the number of the row in the preset table in which the control will zero the rotary axis. The control will make an entry in the **OFFSET** column of the preset table.

Q305 depends on the following parameters:

- **Q337 = 0:** Parameter **Q305** is not effective
- **Q337 = 1:** Parameter **Q305** has the effect described above
- **Q312 = 0:** Parameter **Q305** has the effect described above
- **Q312 > 0:** The entry in **Q305** is ignored. The control will make an entry in the **OFFSET** column, in the row of the preset table that was active when the cycle was called.

Input: **0...99999**

Help graphic**Parameter****Q303 Meas. value transfer (0,1)?**

Define whether the calculated preset will be saved in the datum table or in the preset table:

0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **0, 1**

Q380 Ref. angle in ref. axis?

Angle to which the control will align the probed straight line. Only effective if the rotary axis is in automatic mode or if C is selected (**Q312** = 0 or 6).

Input: **0...360**

Example

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

4.13 Cycle 405 ROT IN C AXIS

ISO programming

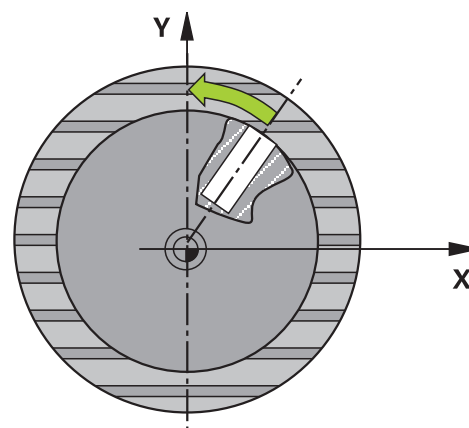
G405

Application

With touch probe cycle **405**, you can measure

- the angular offset between the positive Y axis of the active coordinate system and the center line of a hole
- the angular offset between the nominal position and the actual position of a hole center point

The control compensates for the determined angular offset by rotating the C axis. The workpiece can be clamped in any position on the rotary table, but the Y coordinate of the hole must be positive. If you measure the angular misalignment of the hole with touch probe axis Y (horizontal position of the hole), it may be necessary to execute the cycle more than once because the measuring strategy causes an inaccuracy of approx. 1% of the misalignment.



Instead of Cycle **405 ROT IN C AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

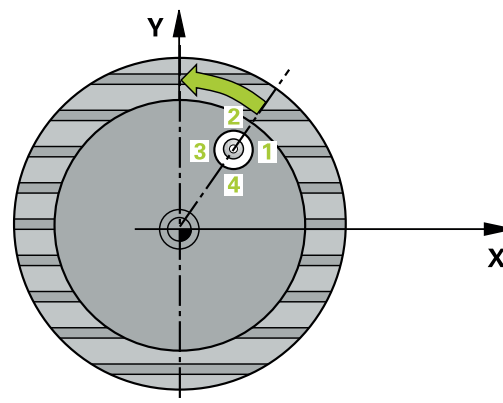
Related topics

- Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES", Page 79

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves along a circular arc, either at measuring height or at clearance height, to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the calculated hole center.
- 5 Finally, the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center, after compensation, lies in the direction of the positive Y axis or at the nominal position of the hole center point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in the parameter **Q150**.



Notes

NOTICE

Danger of collision!

If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There is a risk of collision!

- ▶ The pocket/hole must be free of material on the inside
- ▶ To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

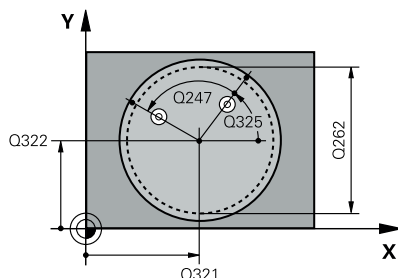
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

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

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the hole in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point with the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point with the nominal position (angle resulting from the position of the hole center). This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large.

Input: **0...99999.9999**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect.

Input: **-120...+120**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

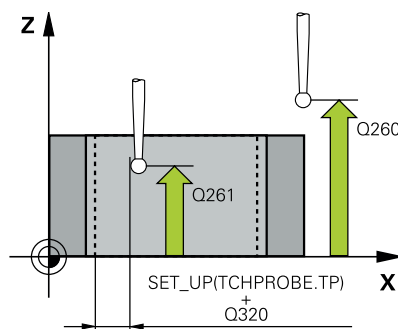
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic**Parameter****Q301 Move to clearance height (0/1)?**

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q337 Set to zero after alignment?

0: Set the display of the C axis to 0 and write to **C_Offset** of the active row of the datum table

> 0: Write the measured angular offset to the datum table. Row number = value in **Q337**. If a C-axis shift is entered in the datum table, the control adds the measured angular offset with the correct sign, positive or negative.

Input: **0...2999**

Example

11 TCH PROBE 405 ROT IN C AXIS ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+10	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+90	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q337=+0	;SET TO ZERO

4.14 Cycle 404 SET BASIC ROTATION

ISO programming

G404

Application

With touch probe cycle **404**, you can set any basic rotation automatically during program run or save it to the preset table. You can also use Cycle **404** if you want to reset an active basic rotation.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

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

Cycle parameters

Help graphic

Parameter

Q307 Preset value for rotation angle

Angle value at which the basic rotation will be set.

Input: **-360.000...+360.000**

Q305 Preset number in table?:

Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter **Q305 = 0** or **Q305 = -1**, the control additionally saves the calculated basic rotation in the basic rotation menu (**Probing rot**) of **Manual Operation** mode.

-1: Overwrite and activate the active preset

0: Copy the active preset to row 0 of the preset table, write the basic rotation to row 0 of the preset table, and activate preset 0

> 1: Save the basic rotation to the specified preset. The preset is not activated.

Input: **-1...99999**

Example

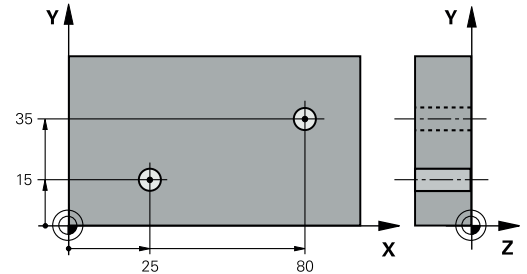
```
11 TCH PROBE 404 SET BASIC ROTATION ~
```

```
Q307=+0 ;PRESET ROTATION ANG. ~
```

```
Q305=-1 ;NUMBER IN TABLE
```


4.15 Example: Determining a basic rotation from two holes

- **Q268** = Center of the 1st hole: X coordinate
- **Q269** = Center of the 1st hole: Y coordinate
- **Q270** = Center of the 2nd hole: X coordinate
- **Q271** = Center of the 2nd hole: Y coordinate
- **Q261** = Coordinate in the touch probe axis in which the measurement is performed
- **Q307** = Angle of the reference line
- **Q402** = Compensation of workpiece misalignment by rotating the table
- **Q337** = Set the display to zero after the alignment



0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 600 Z	
2 TCH PROBE 401 ROT OF 2 HOLES ~	
Q268=+25 ;1ST CENTER 1ST AXIS ~	
Q269=+15 ;1ST CENTER 2ND AXIS ~	
Q270=+80 ;2ND CENTER 1ST AXIS ~	
Q271=+35 ;2ND CENTER 2ND AXIS ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q307=+0 ;PRESET ROTATION ANG. ~	
Q305=+0 ;NUMBER IN TABLE	
Q402=+1 ;COMPENSATION ~	
Q337=+1 ;SET TO ZERO	
3 CALL PGM 35	; Call the part program
4 END PGM TOUCHPROBE MM	

5











**Touch Probe
Cycles: Automatic
Presetting**









5.1 Overview

The control offers cycles for automatic preset setting.



The control must be specifically prepared by the machine manufacturer for the use of a 3D touch probe.
HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

Soft key	Cycle	Page
	Cycle 1400 POSITION PROBING <ul style="list-style-type: none"> ■ Measuring a single position ■ Setting as preset, if applicable 	135
	Cycle 1401 CIRCLE PROBING <ul style="list-style-type: none"> ■ Measuring points on the inside or outside of a circle ■ Setting the circle center as preset, if applicable 	139
	Cycle 1402 SPHERE PROBING <ul style="list-style-type: none"> ■ Measuring points on a sphere ■ Setting the sphere center as preset, if applicable 	144
	Cycle 1404 PROBE SLOT/RIDGE <ul style="list-style-type: none"> ■ Measuring the center of the slot width or ridge width ■ Setting the center as a preset if needed 	149
	Cycle 1430 PROBE POSITION OF UNDERCUT <ul style="list-style-type: none"> ■ Measuring the undercut ■ Measuring the individual position with an L-shaped stylus ■ Definition of preset, if necessary 	153
	Cycle 1434 PROBE SLOT/RIDGE UNDERCUT <ul style="list-style-type: none"> ■ Measuring the undercut ■ Measuring the center of the slot width or ridge width with an L-shaped stylus ■ Setting the center as a preset if needed 	159
	Cycle 410 PRESET INSIDE RECTAN <ul style="list-style-type: none"> ■ Measuring the inside length and width of a rectangle ■ Setting the rectangle center as preset 	167
	Cycle 411 PRESET OUTS. RECTAN <ul style="list-style-type: none"> ■ Measuring the outside length and width of a rectangle ■ Setting the rectangle center as preset 	172
	Cycle 412 PRESET INSIDE CIRCLE <ul style="list-style-type: none"> ■ Measuring any four points on the inside of a circle ■ Setting the circle center as preset 	178
	Cycle 413 PRESET OUTS. CIRCLE <ul style="list-style-type: none"> ■ Measuring any four points on the outside of a circle ■ Setting the circle center as preset 	184

Soft key	Cycle	Page
	Cycle 414 PRESET OUTS. CORNER <ul style="list-style-type: none"> ■ Measuring two straight lines on the outside ■ Setting the intersection of the lines as preset 	190
	Cycle 415 PRESET INSIDE CORNER <ul style="list-style-type: none"> ■ Measuring two straight lines on the inside ■ Setting the intersection of the lines as preset 	196
	Cycle 416 PRESET CIRCLE CENTER <ul style="list-style-type: none"> ■ Measuring any three holes of a bolt hole circle ■ Setting the center of the bolt hole circle as preset 	202
	Cycle 417 PRESET IN TS AXIS <ul style="list-style-type: none"> ■ Measuring any position in the tool axis ■ Setting any position as preset 	208
	Cycle 418 PRESET FROM 4 HOLES <ul style="list-style-type: none"> ■ Measuring two holes on each line crosswise ■ Setting the intersection of the connecting lines as preset 	212
	Cycle 419 PRESET IN ONE AXIS <ul style="list-style-type: none"> ■ Measuring any position in a selectable axis ■ Setting any position in a selectable axis as preset 	217
	Cycle 408 SLOT CENTER PRESET <ul style="list-style-type: none"> ■ Measuring the inside width of a slot ■ Setting the slot center as preset 	221
	Cycle 409 RIDGE CENTER PRESET <ul style="list-style-type: none"> ■ Measuring the outside width of a ridge ■ Setting the ridge center as preset 	226

5.2 Fundamentals of touch probe cycles 14xx for preset setting

Characteristics common to all touch probe cycles 14xx for preset setting

Preset and tool axis

The control sets the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Preset setting in
Z	X and Y
Y	Z and X
X	Y and Z

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q9xx**. You can use the parameters in your NC program. Note the table of result parameters listed with every cycle description.

Programming and operating notes:



- The probing positions are based on the programmed nominal coordinates in the I-CS.
- See your drawing for the nominal positions.
- Before defining a cycle, you must program a tool call in order to define the touch-probe axis.
- The 14xx probing cycles support **SIMPLE** and **L-TYPE** styli.
- In order to achieve optimal accuracy results with an L-TYPE stylus, HEIDENHAIN recommends that you perform probing and calibration at the same speed. Note the setting of the feed override if it is active during probing.

5.3 Cycle 1400 POSITION PROBING

ISO programming

G1400

Application

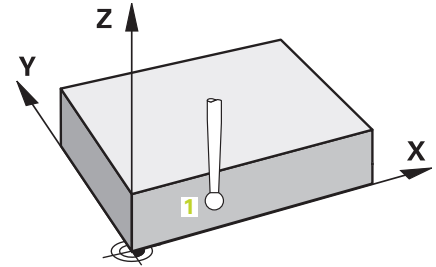
Touch probe cycle **1400** measures any position in a selectable axis. You can apply the result to the active row of the preset table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING",
Page 309

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.
Further information: "Fundamentals of touch probe cycles 14xx for preset setting", Page 134



Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q980 to Q982	Measured deviation from the first touch point
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the first touch point</p>

Notes

NOTICE

Danger of collision!

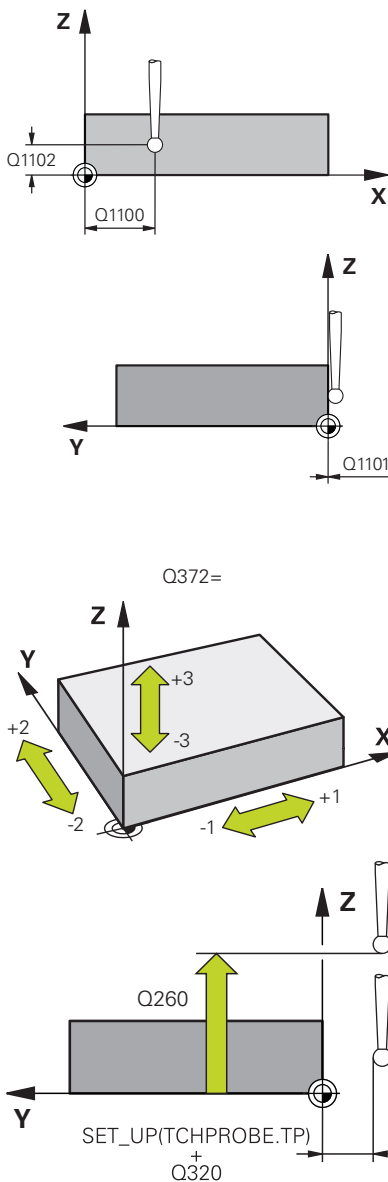
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

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

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the first touch point in the main axis of the working plane

Input: **-99999.9999...+99999.9999** or **?, -, +** or **@**

- **?**: Semi-automatic mode, see Page 55
- **-, +**: Evaluation of the tolerance, see Page 60
- **@**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the first touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q372 Probe direction (-3 to +3)?

Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction.

Input: **-3, -2, -1, +1, +2, +3**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic

Parameter

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

0, 1, 2: Move to the clearance height before and after the touch point. Pre-positioning occurs at **FMAX_PROBE**.

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

Input: **0, 1**

Example

11 TCH PROBE 1400 POSITION PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q372=+0	;PROBING DIRECTION ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

5.4 Cycle 1401 CIRCLE PROBING

ISO programming

G1401

Application

Touch probe cycle **1401** determines the center point of a circular pocket or circular stud. You can transfer the result to the active row of the preset table.

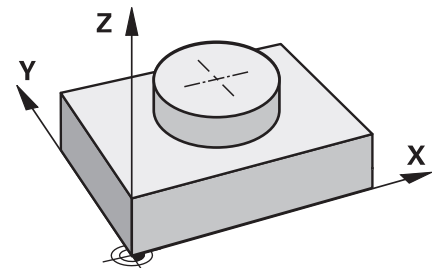
If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 309

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control positions the touch probe to the next touch point.
- 5 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 6 Depending on the definition of **Q423 NO. OF PROBE POINTS**, steps 3 to 5 repeat themselves.
- 7 The control returns the touch probe to the clearance height **Q260**.
- 8 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx for preset setting", Page 134



Q parameter number	Meaning
Q950 to Q952	Measured circle center point in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to Q982	Measured deviation of the circle center
Q996	Measured deviation of the diameter
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from the first circle center</p>
Q973	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation starting from Diameter 1</p>

Notes

NOTICE

Danger of collision!

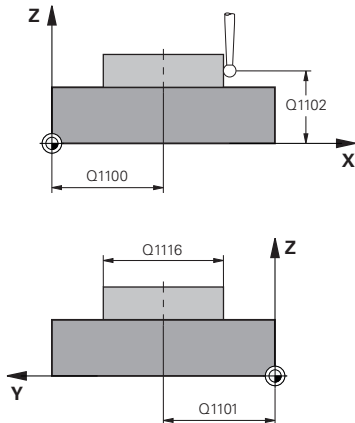
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

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

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the center in the main axis of the working plane.

Input: **-99999.9999...+99999.9999** or enter **?, +, -** or **@**:

- **"?..."**: Semi-automatic mode, see Page 55
- **"...-...+..."**: Evaluation of the tolerance, see Page 60
- **"...@..."**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the center in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1116 Diameter of 1st position?

Diameter of the first hole or the first stud

Input: **0...9999.9999** or optional input:

- **"...-...+..."**: Evaluation of the tolerance, see Page 60

Q1115 Geometry type (0/1)?

Type of object to be probed:

0: Hole

1: Stud

Input: **0, 1**

Q423 Number of probes?

Number of touch points on the diameter

Input: **3, 4, 5, 6, 7, 8**

Q325 Starting angle?

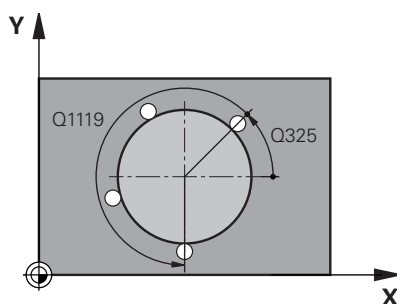
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

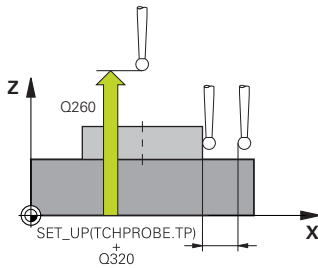
Q1119 Arc angular length?

Angular range in which the touch points are distributed.

Input: **-359.999...+360.000**



Help graphic



Parameter

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points

-1: Do not move to the clearance height.

0, 1: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**.

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

Input: **0, 1**

Example

11 TCH PROBE 1401 CIRCLE PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS1116=+10	;DIAMETER 1 ~
Q1115=+0	;GEOMETRY TYPE ~
Q423=+3	;NO. OF PROBE POINTS ~
Q325=+0	;STARTING ANGLE ~
Q1119=+360	;ANGULAR LENGTH ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

5.5 Cycle 1402 SPHERE PROBING

ISO programming

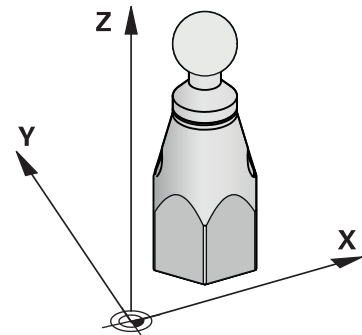
G1402

Application

Touch probe cycle **1402** determines the center point of a sphere. You can apply the result to the active row of the preset table.

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control positions the touch probe to the next touch point.
- 5 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 6 Depending on the definition of **Q423** "Number of probe measurements", steps 3 to 5 repeat themselves.
- 7 The control moves the touch probe in the tool axis by the set-up clearance to a position above the sphere.
- 8 The touch probe moves to the center of the sphere and probes another touch point.
- 9 The touch probe returns to the clearance height **Q260**.
- 10 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.
Further information: "Fundamentals of touch probe cycles 14xx for preset setting", Page 134



Q parameter number	Meaning
Q950 to Q952	Measured circle center in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to Q982	Measured deviation of the circle center
Q996	Measured deviation of the diameter
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>

Notes

NOTICE

Danger of collision!

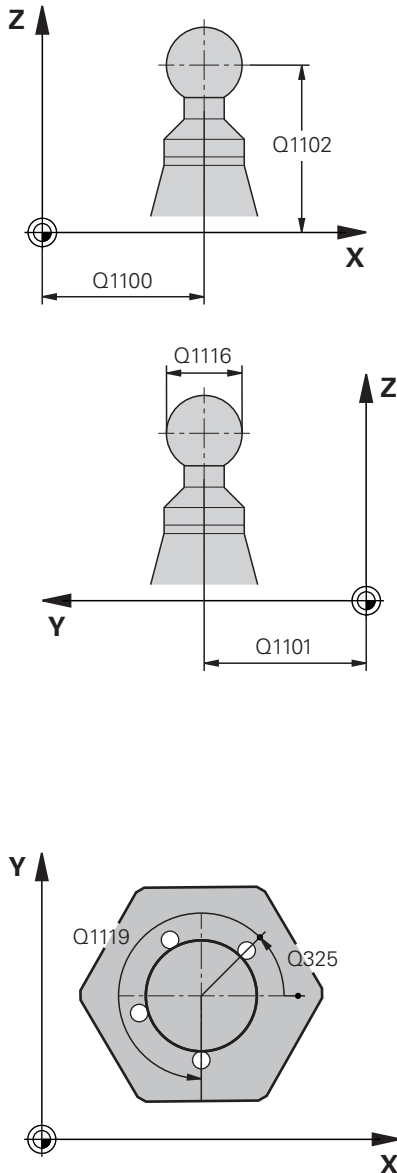
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you have programmed Cycle **1493 EXTRUSION PROBING** before, the control will ignore it during the execution of Cycle **1402 SPHERE PROBING**.

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the center in the main axis of the working plane.

Input: **-99999.9999...+99999.9999** or enter **?, +, -** or **@**:

- **"?..."**: Semi-automatic mode, see Page 55
- **"...-...+..."**: Evaluation of the tolerance, see Page 60
- **"...@..."**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the center in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1116 Diameter of 1st position?

Diameter of the sphere

Input: **0...9999.9999** or optional input (see **Q1100**)

- **"...-...+..."**: Evaluation of the tolerance, see Page 60

Q423 Number of probes?

Number of touch points on the diameter

Input: **3, 4, 5, 6, 7, 8**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q1119 Arc angular length?

Angular range in which the touch points are distributed.

Input: **-359.999...+360.000**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Help graphic**Parameter****Q260 Clearance height?**

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points

-1: Do not move to the clearance height.

0, 1: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**.

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction of the active preset based on the center of the sphere. The control corrects the active present by the amount of the deviation of the nominal and actual position of the center.

Input: **0, 1**

Example

11 TCH PROBE 1402 SPHERE PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS1116=+10	;DIAMETER 1 ~
Q423=+3	;NO. OF PROBE POINTS ~
Q325=+0	;STARTING ANGLE ~
Q1119=+360	;ANGULAR LENGTH ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

5.6 Cycle 1404 PROBE SLOT/RIDGE

ISO programming

G1404

Application

Touch probe cycle **1404** determines the center of the width of a slot or ridge. The control probes the two opposing touch points. The control probes perpendicularly to the angle of rotation of the object to be probed, even if the object to be probed is rotated. You can apply the result to the active row of the preset table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 309

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

Further information: "Positioning logic", Page 45

- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.

- 3 Depending on the selected type of geometry in the parameter **Q1115**, the control proceeds as follows:

Slot **Q1115=0**:

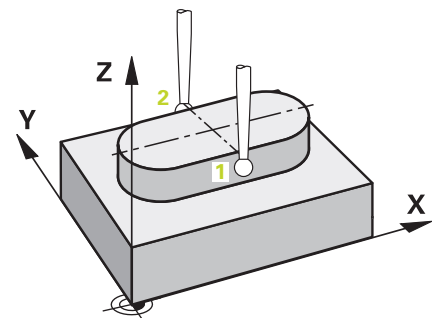
- If you program **CLEAR. HEIGHT MODE Q1125** with the value **0, 1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to **Q260 CLEARANCE HEIGHT**.

Ridge **Q1115=1**:

- Independently of **Q1125**, the control positions the touch probe at **FMAX_PROBE** after every touch point back to **Q260 CLEARANCE HEIGHT**.

- 4 The touch probe moves to the next touch point **2** and performs the second probing procedure at the probing rate **F**.
- 5 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx for preset setting", Page 134



Q parameter number	Meaning
Q950 to Q952	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to Q982	Measured deviation of the center of the slot or ridge
Q998	Measured deviation of the slot width or ridge width
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation from the center of the slot or ridge</p>
Q975	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation based on the slot width or ridge width</p>

Notes

NOTICE

Danger of collision!

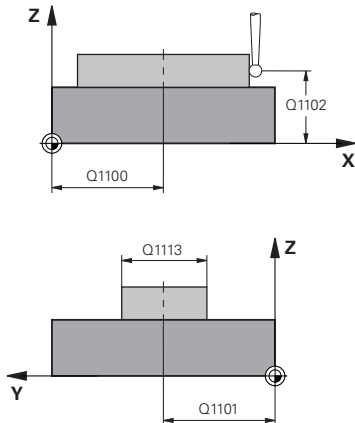
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

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

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the center in the main axis of the working plane.

Input: **-99999.9999...+99999.9999** or enter **?, +, -** or **@**:

- **"?..."**: Semi-automatic mode, see Page 55
- **"...-...+..."**: Evaluation of the tolerance, see Page 60
- **"...@..."**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the center in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the touch points in the tool axis

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1113 Width of slot/ridge?

Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect.

Input: **0...9999.9999** Or - or +:

- **"...-...+..."**: Evaluation of the tolerance, see Page 60

Q1115 Geometry type (0/1)?

Type of object to be probed:

0: Slot

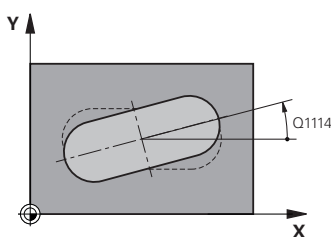
1: Ridge

Input: **0, 1**

Q1114 Angle of rotation?

Angle about which the slot or the ridge is rotated. The center of rotation is in **Q1100** and **Q1101**. This value has an absolute effect.

Input: **0...359999**



Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

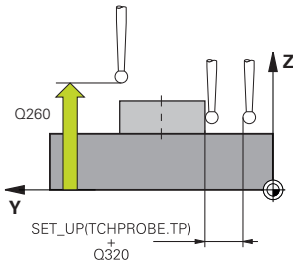
Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic



Parameter

Q1125 Traverse to clearance height?

Positioning behavior between the touch points with a slot:

-1: Do not move to the clearance height.

0, 1: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**.

The parameter takes effect only with **Q1115=+1** (slot).

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction of the active preset based on the center of the slot or the ridge. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the center.

Input: **0, 1**

Example

11 TCH PROBE 1404 PROBE SLOT/RIDGE ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1113=+20	;WIDTH OF SLOT/RIDGE ~
Q1115=+0	;GEOMETRY TYPE ~
Q1114=+0	;ANGLE OF ROTATION ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

5.7 Cycle 1430 PROBE POSITION OF UNDERCUT

ISO programming

G1430

Application

Touch probe cycle **1430** allows a position to be probed with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. You can apply the result of the probing procedure to the active rows of the preset table.

In the main axis and secondary axis, the touch probe is oriented in accordance with the calibration angle. In the tool axis, the touch probe is oriented in accordance with the programmed spindle angle and the calibration angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING",
Page 309

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

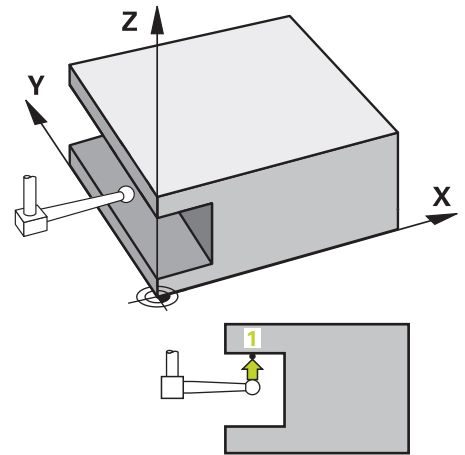
Pre-position in the machining plane based on the probing direction:

- **Q372=+/-1**: The pre-position in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position **Q1100**. The radial approach length takes effect in the direction opposite to the probing direction.
- **Q372=+/-2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the **Q1101**. The radial approach length takes effect in the direction opposite to the probing direction.
- **Q372=+/-3**: The pre-position of the main axis and secondary axis depends on the direction in which the stylus is oriented. The pre-position is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position. The radial approach length takes effect in the direction opposite to the spindle angle **Q336**.

Further information: "Positioning logic", Page 45

- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.
- 3 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
- 4 If you program **CLEAR. HEIGHT MODE Q1125** with the value **0, 1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 5 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx for preset setting", Page 134



Q parameter number	Meaning
Q950 to Q952	Measured position in the main axis, auxiliary axis and tool axis
Q980 to Q982	Measured deviation of the position in the main axis, auxiliary axis and tool axis
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation based on the nominal position of the first touch point</p>

Notes

NOTICE

Danger of collision!

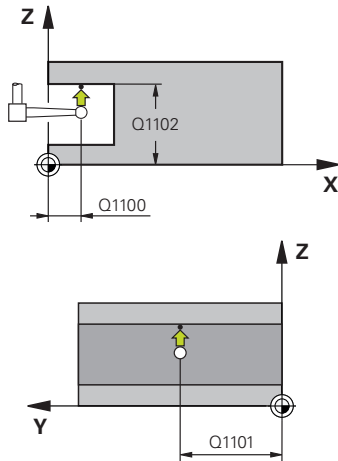
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle is not intended for L-shaped styli. For simple styli, HEIDENHAIN recommends Cycle **1400 POSITION PROBING**.
Further information: "Cycle 1400 POSITION PROBING", Page 135

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the first touch point in the main axis of the working plane

Input: **-99999.9999...+99999.9999** or **?, -, +** or **@**

- **?**: Semi-automatic mode, see Page 55
- **-, +**: Evaluation of the tolerance, see Page 60
- **@**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the first touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q372 Probe direction (-3 to +3)?

Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction.

Input: **-3, -2, -1, +1, +2, +3**

Q336 Angle for spindle orientation?

Angle at which the control orients the tool prior to the probing procedure. This angle takes effect only during probing in the tool axis (**Q372 = +/- 3**). This value has an absolute effect.

Input: **0...360**

Q1118 Distance of radial approach?

Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing.

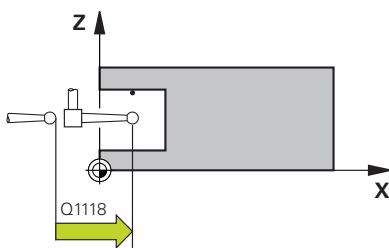
If **Q372 = +/- 1**: Distance is in the direction opposite to the probing direction.

If **Q372 = +/- 2**: Distance is in the direction opposite to the probing direction.

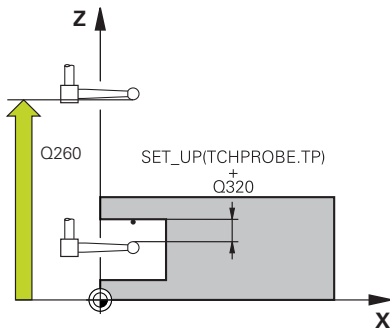
If **Q372 = +/- 3**: Distance is in the direction opposite to the angle of the spindle **Q336**.

This value has an incremental effect.

Input: **0...9999.9999**



Help graphic



Parameter

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

0, 1, 2: Move to the clearance height before and after the touch point. Pre-positioning occurs at **FMAX_PROBE**.

Input: **-1, 0, +1, +2**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

Input: **0, 1**

Example

11 TCH PROBE 1430 PROBE POSITION OF UNDERCUT ~	
Q1100=+10	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-15	;1ST POINT TOOL AXIS ~
Q372=+1	;PROBING DIRECTION ~
Q336=+0	;ANGLE OF SPINDLE ~
Q1118=+20	;RADIAL APPROACH PATH ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

5.8 Cycle 1434 PROBE SLOT/RIDGE UNDERCUT

ISO programming
G1434

Application

Touch probe cycle **1434** determines the center and width of a slot or a ridge using an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes the two opposing touch points. You can apply the result to the active row of the preset table.

The control orients the touch probe to the calibration angle from the touch probe table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING",
Page 309

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

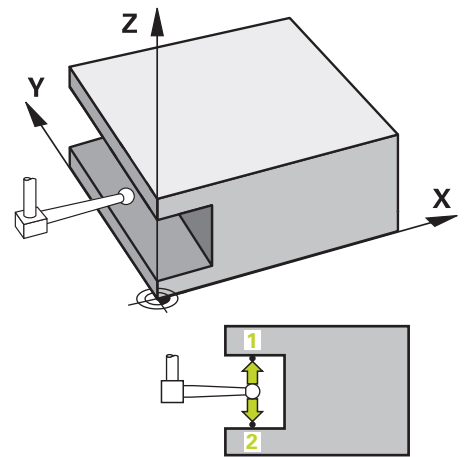
The pre-position in the machining plane depends on the object plane:

- **Q1139=+1**: The pre-position in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position in **Q1100**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the secondary axis is equivalent to the nominal position.
- **Q1139=+2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position in **Q1101**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the main axis is equivalent to the nominal position.

Further information: "Positioning logic", Page 45

- 2 The control then positions the touch probe at the entered measuring height **Q1102** and performs the first probing procedure **1** at probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.
- 3 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
- 4 The control positions the touch probe to the next touch point **2** and performs the second probing procedure at probing feed rate **F**.
- 5 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
- 6 If you program the parameter **CLEAR. HEIGHT MODE Q1125** with the value **0** or **1**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 7 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx for preset setting", Page 134



Q parameter number	Meaning
Q950 to Q952	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to Q982	Measured deviation of the center of the slot or ridge
Q998	Measured deviation of the slot width or ridge width
Q183	<p>Workpiece status</p> <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not moved. <p>The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle.</p> <p>Further information: "Cycle 441 FAST PROBING", Page 306</p>
Q970	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation based on the center of the slot or the ridge</p>
Q975	<p>If you have programmed Cycle 1493 EXTRUSION PROBING:</p> <p>Maximum deviation based on the slot width or ridge width</p>

Notes

NOTICE

Danger of collision!

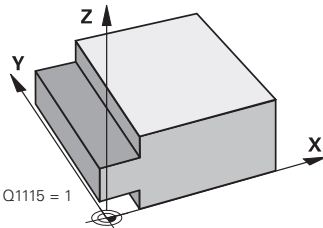
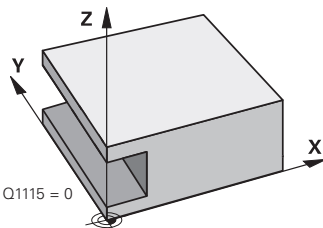
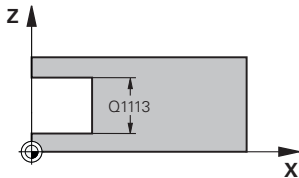
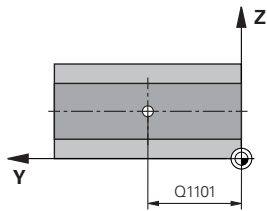
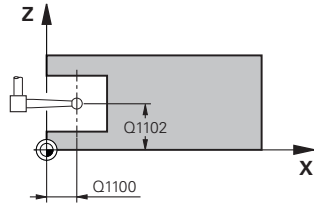
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
 - If you program in the radial approach length **Q1118=-0**, then the algebraic sign has no effect. The behavior is identical to +0.
 - This cycle is intended for an L-shaped stylus. For simple styli, HEIDENHAIN recommends Cycle **1404 PROBE SLOT/RIDGE**.
- Further information:** "Cycle 1404 PROBE SLOT/RIDGE", Page 149

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the center in the main axis of the working plane.

Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**:

- **"?..."**: Semi-automatic mode, see Page 55
- **"...-...+..."**: Evaluation of the tolerance, see Page 60
- **"...@..."**: Transfer of an actual position, see Page 63

Q1101 1st noml. position of minor axis?

Absolute nominal position of the center in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute spindle position of the center in the tool axis

Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)

Q1113 Width of slot/ridge?

Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect.

Input: **0...9999.9999** Or - or +:

"...-...+...": Evaluation of the tolerance, see Page 60

Q1115 Geometry type (0/1)?

Type of object to be probed:

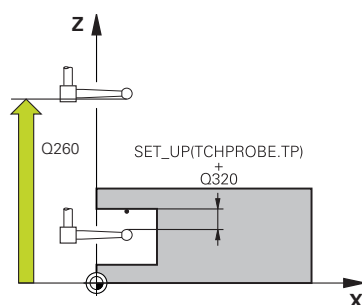
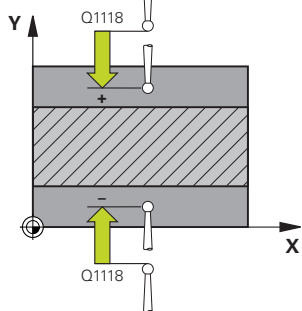
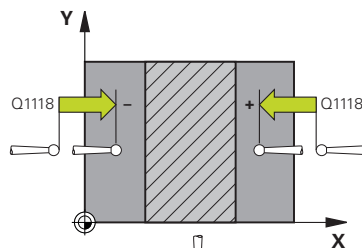
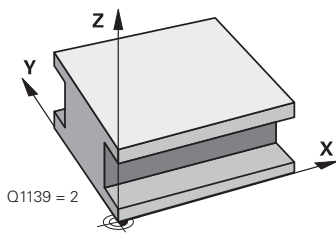
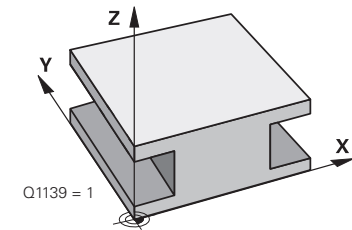
0: Slot

1: Ridge

Input: **0, 1**

Help graphic

Parameter

**Q1139 Object plane (1-2)?**

Plane in which the control interprets the probing direction.

1: YZ plane

2: ZX plane

Input: **1, 2**

Q1118 Distance of radial approach?

Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing. The direction of **Q1118** is equivalent to the probing direction and is in the direction opposite to the algebraic sign. This value has an incremental effect.

Input: **-9999.9999...+9999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q1125 Traverse to clearance height?

Positioning behavior before and after the cycle:

-1: Do not move to the clearance height.

0, 1: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

Input: **-1, 0, +1**

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.

1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.

2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0, 1, 2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction of the active preset based on the center of the slot or the ridge. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the center.

Input: **0, 1**

Example

11 TCH PROBE 1434 PROBE SLOT/RIDGE UNDERCUT ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1113=+20	;WIDTH OF SLOT/RIDGE ~
Q1115=+0	;GEOMETRY TYPE ~
Q1139=+1	;OBJECT PLANE ~
Q1118=-15	;RADIAL APPROACH PATH ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

5.9 Fundamentals of touch probe cycles 408 to 419 for preset setting

Application



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

The control offers cycles for automatically determining presets and handling them as follows:

- Setting the calculated values directly as display values
- Writing the calculated values to the preset table
- Writing the calculated values to a datum table

Preset and touch probe axis

The control determines the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Set preset in
Z	X and Y
Y	Z and X
X	Y and Z

Saving the calculated preset

In all cycles for presetting, you can use input parameters **Q303** and **Q305** to define how the control is to save the calculated preset:

- **Q305 = 0, Q303 = 1:**
The control copies the active preset to row 0, changes it and activates row 0, deleting simple transformations.
- **Q305 not equal to 0, Q303 = 0:**
The result is written to the datum table, row **Q305**; **activate the datum with TRANS DATUM in the NC program**
Further information: Klartext Programming User's Manual
- **Q305 not equal to 0, Q303 = 1:**
The result is written to the preset table, row **Q305**; **use Cycle 247 to activate the preset in the NC program**
- **Q305 not equal to 0, Q303 = -1**



This combination can only occur if you

- read in NC programs (containing Cycles **410** to **418**) that were created on a TNC 4xx
- read in NC programs (containing Cycles **410** to **418**) that were created with an older software version of an iTNC 530
- did not specifically define the measured-value transfer with parameter **Q303** when defining the cycle

In these cases, the control outputs an error message, since the complete handling of REF-referenced datum tables has changed. You must define a measured-value transfer yourself with parameter **Q303**.

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. You can use these parameters in your NC program. Note the table of result parameters listed with every cycle description.

5.10 Cycle 410 PRESET INSIDE RECTAN

ISO programming

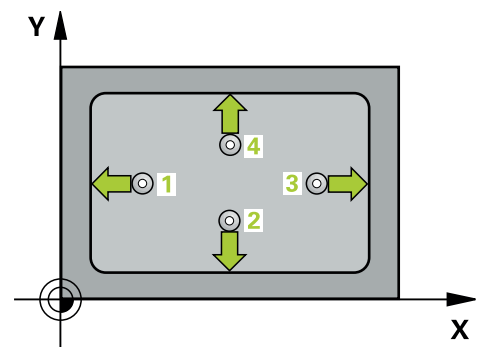
G410

Application

Touch probe cycle **410** finds the center of a rectangular pocket and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 7 Then the control saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE

Danger of collision!

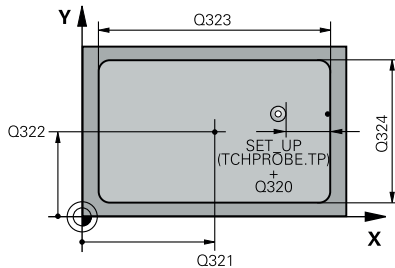
If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There is a risk of collision!

- ▶ To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides.
- ▶ Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the pocket in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the pocket in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q323 First side length?

Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q324 Second side length?

Pocket length, parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

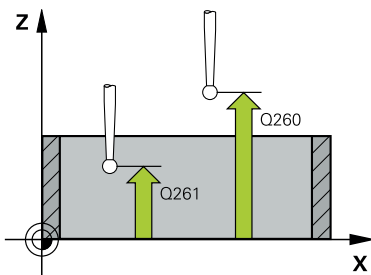
Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**



Help graphic

Parameter

Q305 Number in table?

Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303=1**, the control will write the data to the preset table.

If **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Help graphic

Parameter

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Example

11 CYCL DEF 410 PRESET INSIDE RECTAN ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q323=+60	;FIRST SIDE LENGTH ~
Q324=+20	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+10	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.11 Cycle 411 PRESET OUTS. RECTAN

ISO programming

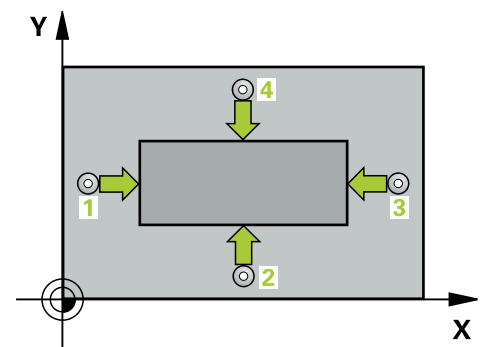
G411

Application

Touch probe cycle **411** finds the center of a rectangular stud and defines this position as the datum. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 7 Then the control saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE

Danger of collision!

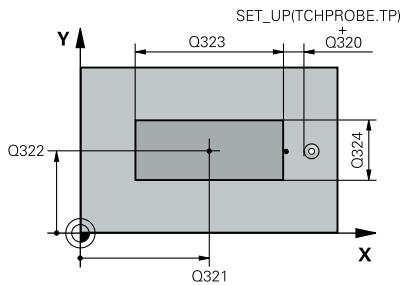
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the stud in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the stud in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q323 First side length?

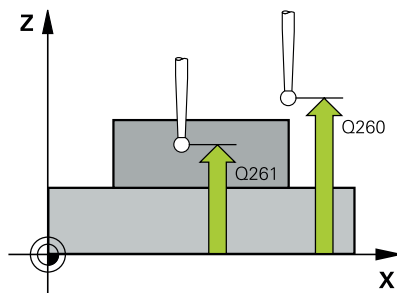
Length of stud parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q324 Second side length?

Length of stud parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Help graphic**Parameter****Q305 Number in table?**

Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303=1**, the control will write the data to the preset table.

If **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Help graphic

Parameter

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 411 PRESET OUTS. RECTAN ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q323=+60	;FIRST SIDE LENGTH ~
Q324=+20	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+0	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.12 Cycle 412 PRESET INSIDE CIRCLE

ISO programming

G412

Application

Touch probe cycle **412** finds the center of a circular pocket (hole) and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



Instead of Cycle **412 PRESET INSIDE CIRCLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

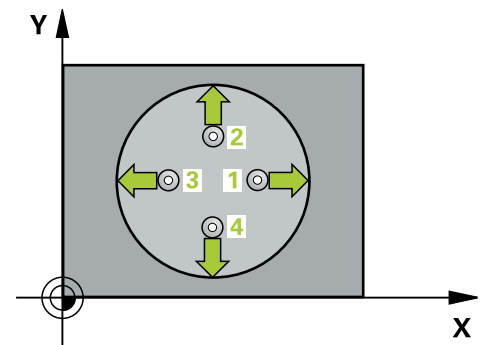
Related topics

- Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING", Page 139

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or linearly at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 7 Then the control saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE

Danger of collision!

If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There is a risk of collision!

- ▶ The pocket/hole must be free of material on the inside
- ▶ To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

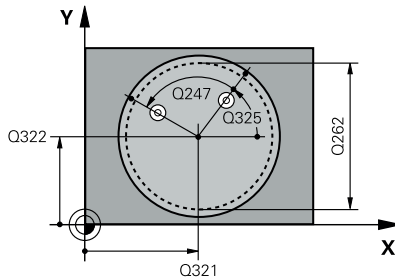
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°



Program the stepping angle to be less than 90°

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the pocket in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the pocket in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large.

Input: **0...99999.9999**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect.

Input: **-120...+120**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

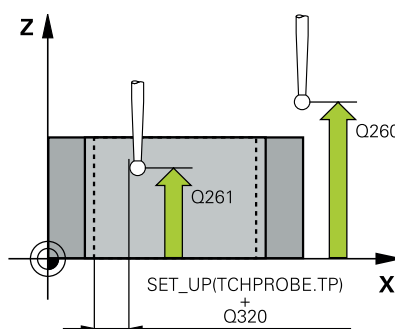
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic

Parameter

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q305 Number in table?

Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303=1**, the control will write the data to the preset table.

If **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Help graphic

Parameter

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q423 No. probe points in plane (4/3)?

Define whether the control will use three or four touch points to measure the circle:

3: Use three measuring points

4: Use four measuring points (default setting)

Input: **3, 4**

Q365 Type of traverse? Line=0/arc=1

Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active.

0: Move in a straight line between machining operations

1: Move along a circular arc on the pitch circle diameter between machining operations

Input: **0, 1**

Example

11 TCH PROBE 412 PRESET INSIDE CIRCLE ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+12	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE

5.13 Cycle 413 PRESET OUTS. CIRCLE

ISO programming

G413

Application

Touch probe cycle **413** finds the center of a circular stud and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



Instead of Cycle **413 PRESET OUTS. CIRCLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

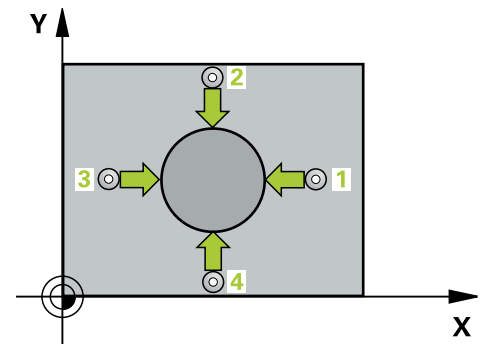
Related topics

- Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING", Page 139

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Application", Page 165
- 7 Then the control saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q151	Actual value of center in main axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE

Danger of collision!

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

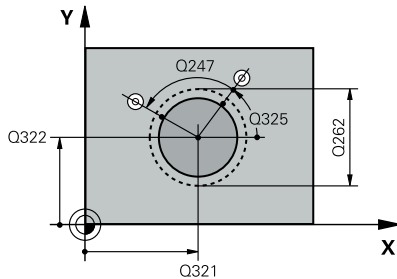
- ▶ Before a cycle definition you must have programmed a tool call to define the touch probe axis.
- The control will reset an active basic rotation at the beginning of the cycle.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°



Program the stepping angle to be less than 90°

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the stud in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+9999.9999**

Q322 Center in 2nd axis?

Center of the stud in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Approximate diameter of the stud. Enter a value that is more likely to be too large than too small.

Input: **0...99999.9999**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect.

Input: **-120...+120**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

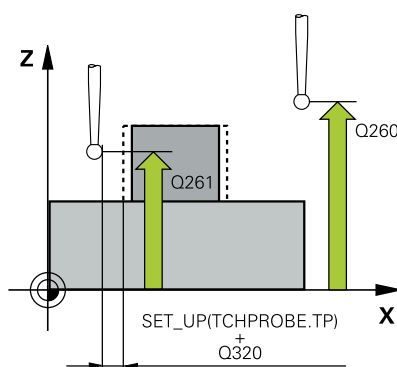
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic**Parameter****Q301 Move to clearance height (0/1)?**

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q305 Number in table?

Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303=1**, the control will write the data to the preset table.

If **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Help graphic

Parameter

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q423 No. probe points in plane (4/3)?

Define whether the control will use three or four touch points to measure the circle:

3: Use three measuring points

4: Use four measuring points (default setting)

Input: **3, 4**

Q365 Type of traverse? Line=0/arc=1

Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active.

0: Move in a straight line between machining operations

1: Move along a circular arc on the pitch circle diameter between machining operations

Input: **0, 1**

Example

11 TCH PROBE 413 PRESET OUTS. CIRCLE ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+15	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE

5.14 Cycle 414 PRESET OUTS. CORNER

ISO programming

G414

Application

Touch probe cycle **414** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

i Instead of Cycle **414 PRESET OUTS. CORNER**, HEIDENHAIN recommends using the more powerful Cycle **1416 INTERSECTION PROBING**.

Related topics

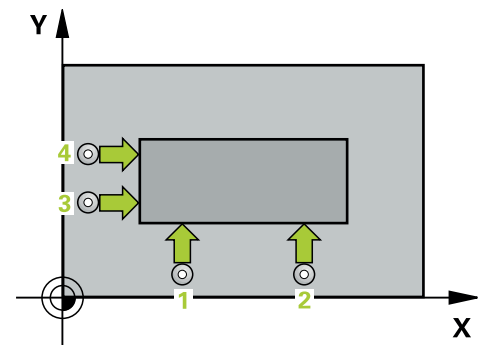
- Cycle **1416 INTERSECTION PROBING**

Further information: "Cycle 1416 INTERSECTION PROBING", Page 95

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 7 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
- 8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

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



Q parameter number	Meaning
--------------------	---------

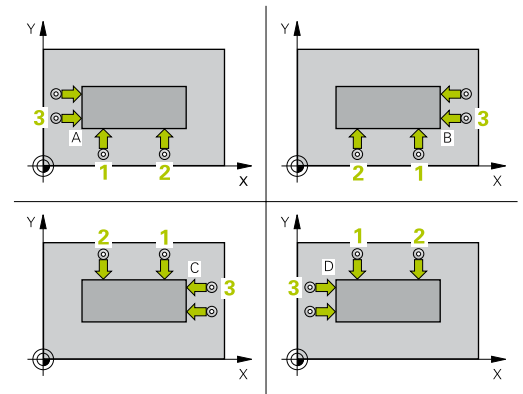
Q151	Actual value of corner in reference axis
------	--

Q152	Actual value of corner in minor axis
------	--------------------------------------

Definition of the corner

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

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

**Notes****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

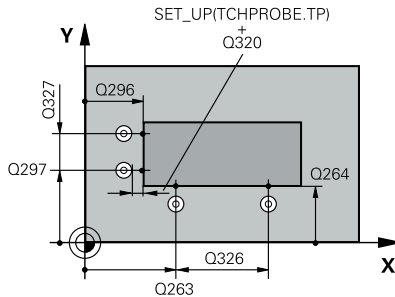
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q326 Spacing in 1st axis?

Distance between the first and second measuring points in the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q296 3rd measuring point in 1st axis?

Coordinate of the third touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q297 3rd measuring point in 2nd axis?

Coordinate of the third touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q327 Spacing in 2nd axis?

Distance between third and fourth measuring points in the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

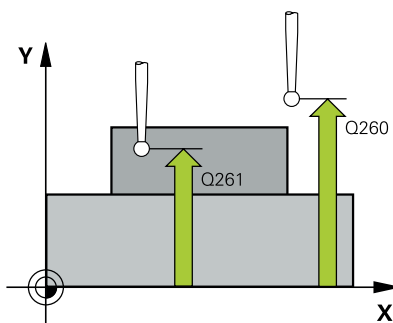
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**



Help graphic**Parameter****Q260 Clearance height?**

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q304 Execute basic rotation (0/1)?

Define whether the control will compensate workpiece misalignment with a basic rotation:

0: No basic rotation

1: Basic rotation

Input: **0, 1**

Q305 Number in table?

Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Help graphic

Parameter

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 414 PRESET OUTS. CORNER ~	
Q263=+37	;1ST POINT 1ST AXIS ~
Q264=+7	;1ST POINT 2ND AXIS ~
Q326=+50	;SPACING IN 1ST AXIS ~
Q296=+95	;3RD PNT IN 1ST AXIS ~
Q297=+25	;3RD PNT IN 2ND AXIS ~
Q327=+45	;SPACING IN 2ND AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q304=+0	;BASIC ROTATION ~
Q305=+7	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.15 Cycle 415 PRESET INSIDE CORNER

ISO programming

G415

Application

Touch probe cycle **415** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

i Instead of Cycle **415 PRESET INSIDE CORNER**, HEIDENHAIN recommends using the more powerful Cycle **1416 INTERSECTION PROBING**.

Related topics

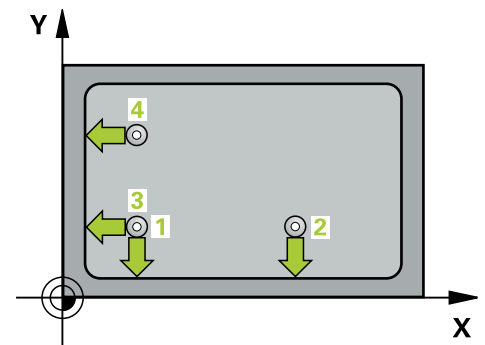
- Cycle **1416 INTERSECTION PROBING**

Further information: "Cycle 1416 INTERSECTION PROBING", Page 95

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The probing direction is derived from the number by which you identify the corner.
- 3 The touch probe moves to the next touch point **2**; the control offsets the touch probe in the secondary axis by the amount of the set-up clearance **Q320 + SET_UP** + ball-tip radius and then performs the second probing operation
- 4 The control positions the touch probe at touch point **3** (same positioning logic as for the first touch point) and performs the probing operation there
- 5 The touch probe then moves to touch point **4**. The control offsets the touch probe in the main axis by the amount of the set-up clearance **Q320 + SET_UP** + ball-tip radius and then performs the fourth probing operation
- 6 The control returns the touch probe to the clearance height.
- 7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 8 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
- 9 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

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



Q parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

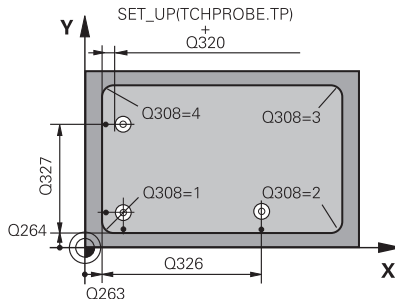
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the corner in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the corner in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q326 Spacing in 1st axis?

Distance between the first corner and the second measuring point in the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q327 Spacing in 2nd axis?

Distance between the corner and the fourth measuring point in the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q308 Corner? (1/2/3/4)

Number identifying the corner at which the control will set the preset.

Input: **1, 2, 3, 4**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

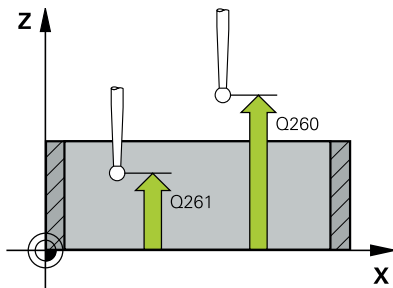
Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**



Help graphic

Parameter

Q304 Execute basic rotation (0/1)?

Define whether the control will compensate workpiece misalignment with a basic rotation:

0: No basic rotation

1: Basic rotation

Input: **0, 1**

Q305 Number in table?

Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Help graphic

Parameter

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 415 PRESET INSIDE CORNER ~	
Q263=+37	;1ST POINT 1ST AXIS ~
Q264=+7	;1ST POINT 2ND AXIS ~
Q326=+50	;SPACING IN 1ST AXIS ~
Q327=+45	;SPACING IN 2ND AXIS ~
Q308=+1	;CORNER ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q304=+0	;BASIC ROTATION ~
Q305=+7	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.16 Cycle 416 PRESET CIRCLE CENTER

ISO programming

G416

Application

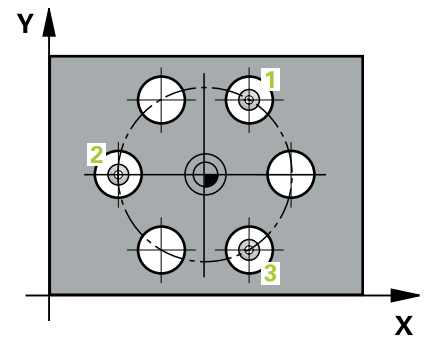
Touch probe cycle **416** finds the center of a bolt hole circle by measuring three holes, and defines the determined center as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle run

- 1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic

Further information: "Positioning logic", Page 45

- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
- 6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
- 7 The control returns the touch probe to the clearance height.
- 8 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 9 Then the control saves the actual values in the Q parameters listed below.
- 10 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

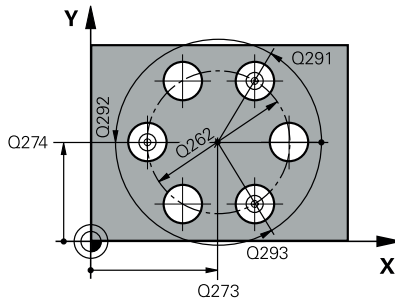
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Bolt hole circle center (nominal value) in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Bolt hole circle center (nominal value) in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be.

Input: **0...99999.9999**

Q291 Polar coord. angle of 1st hole?

Polar coordinate angle of the first hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q292 Polar coord. angle of 2nd hole?

Polar coordinate angle of the second hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q293 Polar coord. angle of 3rd hole?

Polar coordinate angle of the third hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic**Parameter****Q305 Number in table?**

Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303=1**, the control will write the data to the preset table.

If **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated bolt-hole center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated bolt-hole circle center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Help graphic

Parameter

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Example

11 TCH PROBE 416 PRESET CIRCLE CENTER ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+90	;NOMINAL DIAMETER ~
Q291=+34	;ANGLE OF 1ST HOLE ~
Q292=+70	;ANGLE OF 2ND HOLE ~
Q293=+210	;ANGLE OF 3RD HOLE ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+20	;CLEARANCE HEIGHT ~
Q305=+12	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET ~
Q320=+0	;SET-UP CLEARANCE

5.17 Cycle 417 PRESET IN TS AXIS

ISO programming

G417

Application

Touch probe cycle **417** measures any coordinate in the touch probe axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.



Instead of Cycle **417 PRESET IN TS AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

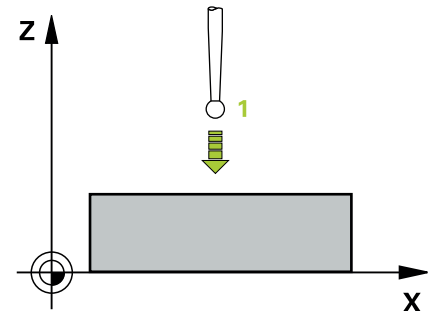
Related topics

- Cycle **1400 POSITION PROBING**

Further information: "Cycle 1400 POSITION PROBING", Page 135

Cycle run

- 1 Following the positioning logic, the control positions the touch probe to the programmed touch point **1**. In this process, the control offsets the touch probe by the set-up clearance in the direction of the positive touch probe axis.
- Further information:** "Positioning logic", Page 45
- 2 Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement
 - 3 The control returns the touch probe to the clearance height.
 - 4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
 - 5 Then the control saves the actual values in the Q parameters listed below.



Q parameter number	Meaning
Q160	Actual value of measured point

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

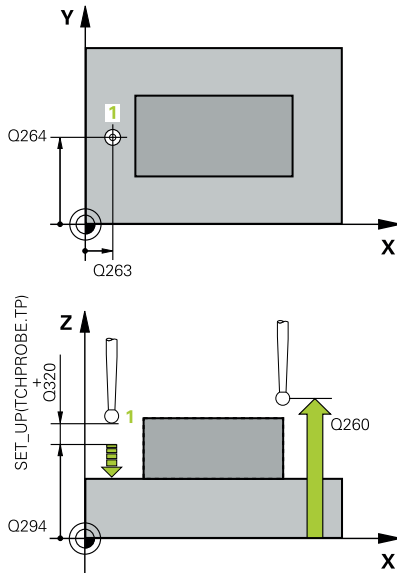
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control sets the preset in this axis.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q294 1st measuring point in 3rd axis?

Coordinate of the first touch point in the touch probe axis. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q305 Number in table?

Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Help graphic**Parameter****Q303 Meas. value transfer (0,1)?**

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Example

11 TCH PROBE 417 PRESET IN TS AXIS ~	
Q263=+25	;1ST POINT 1ST AXIS ~
Q264=+25	;1ST POINT 2ND AXIS ~
Q294=+25	;1ST POINT 3RD AXIS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q305=+0	;NUMBER IN TABLE ~
Q333=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER

5.18 Cycle 418 PRESET FROM 4 HOLES

ISO programming

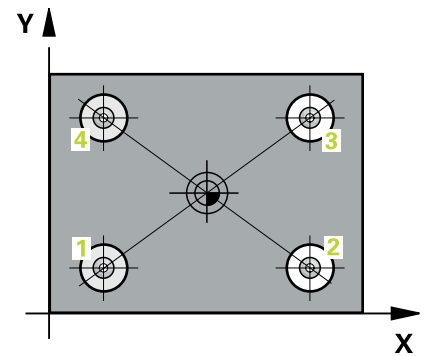
G418

Application

Touch probe cycle **418** calculates the intersection of the lines connecting two opposite hole center points and sets the preset at the point of intersection. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle run

- 1 The control positions the touch probe at the center of the first hole **1**, using positioning logic
Further information: "Positioning logic", Page 45
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The control repeats this step for holes **3** and **4**.
- 6 The control returns the touch probe to the clearance height.
- 7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 8 The control calculates the preset as the intersection of the lines connecting the centers of holes **1/3** and **2/4** and saves the actual values in the Q parameters listed below.
- 9 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

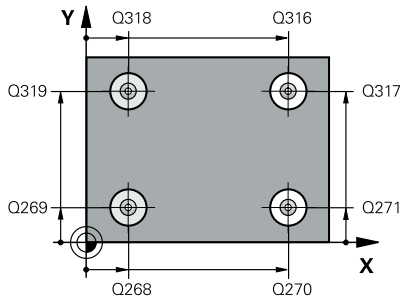
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q268 1st hole: center in 1st axis?

Center of the first hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q269 1st hole: center in 2nd axis?

Center of the first hole in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q270 2nd hole: center in 1st axis?

Center of the second hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q271 2nd hole: center in 2nd axis?

Center of the second hole in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q316 3rd hole: Center in 1st axis?

Center of the third hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q317 3rd hole: Center in 2nd axis?

Center of the third hole in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q318 4th hole: Center in 1st axis?

Center of the fourth hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q319 4th hole: Center in 2nd axis?

Center of the fourth hole in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

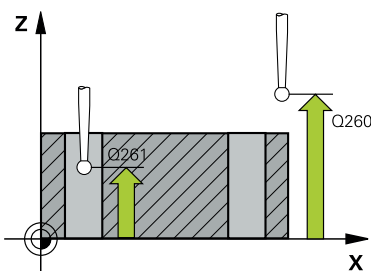
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic

Parameter

Q305 Number in table?

Indicate the number of the row in the preset table or datum table in which the control saves the coordinates of the point of intersection of the connecting lines. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Help graphic

Parameter

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 418 PRESET FROM 4 HOLES ~	
Q268=+20	;1ST CENTER 1ST AXIS ~
Q269=+25	;1ST CENTER 2ND AXIS ~
Q270=+150	;2ND CENTER 1ST AXIS ~
Q271=+25	;2ND CENTER 2ND AXIS ~
Q316=+150	;3RD CENTER 1ST AXIS ~
Q317=+85	;3RD CENTER 2ND AXIS ~
Q318=+22	;4TH CENTER 1ST AXIS ~
Q319=+80	;4TH CENTER 2ND AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q305=+12	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+0	;PRESET

5.19 Cycle 419 PRESET IN ONE AXIS

ISO programming

G419

Application

Touch probe cycle **419** measures any coordinate in the a selectable axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.



Instead of Cycle **419 PRESET IN ONE AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

Further information: "Cycle 1400 POSITION PROBING", Page 135

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
- 3 The control returns the touch probe to the clearance height.
- 4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Application", Page 165

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

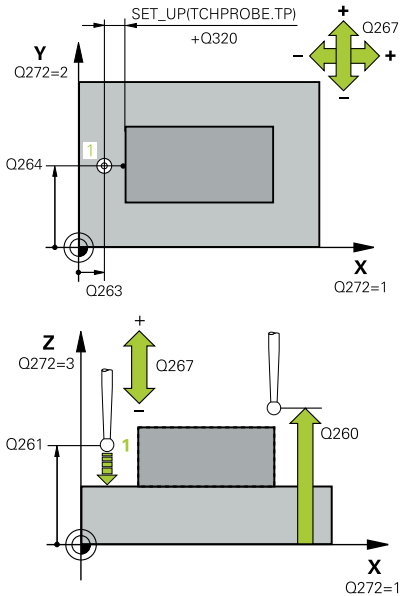
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you want to save the preset in several axes in the preset table, you can use Cycle **419** several times in a row. However, you also have to reactivate the preset number after every run of Cycle **419**. If you work with preset 0 as active preset, this process is not required.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis
- 3: Touch probe axis = measuring axis

Axis assignment

Active touch probe axis: Q272 = 3	Corresponding main axis: Q272 = 1	Corresponding secondary axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

Input: **1, 2, 3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

- 1: Negative traverse direction
- +1: Positive traverse direction

Input: **-1, +1**

Help graphic

Parameter

Q305 Number in table?

Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q333 New preset?

Coordinate at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 165

0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **-1, 0, +1**

Example

11 TCH PROBE 419 PRESET IN ONE AXIS ~	
Q263=+25	;1ST POINT 1ST AXIS ~
Q264=+25	;1ST POINT 2ND AXIS ~
Q261=+25	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q272=+1	;MEASURING AXIS ~
Q267=+1	;TRAVERSE DIRECTION ~
Q305=+0	;NUMBER IN TABLE ~
Q333=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER

5.20 Cycle 408 SLOT CENTER PRESET

ISO programming

G408

Application

Touch probe cycle **408** finds the center of a slot and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



Instead of Cycle **408 SLOT CENTER PRESET**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

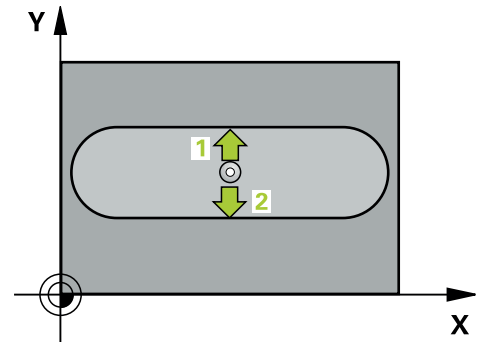
Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE", Page 149

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height.
- 5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 6 Then the control saves the actual values in the Q parameters listed below.
- 7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Notes

NOTICE**Danger of collision!**

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE**Danger of collision!**

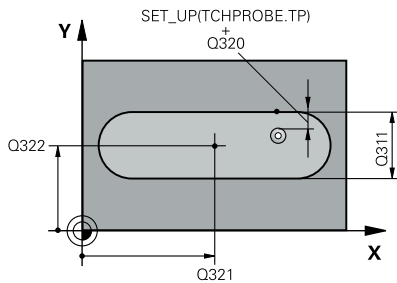
If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch probe does not return to the clearance height between the two measuring points. There is a risk of collision!

- ▶ To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width.
- ▶ Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the slot in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the slot in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q311 Width of slot?

Width of the slot, regardless of its position in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis

Input: **1, 2**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

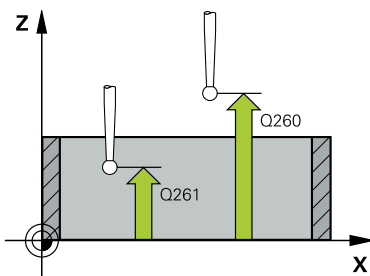
Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

- 0: Move to measuring height between measuring points
- 1: Move to clearance height between measuring points

Input: **0, 1**



Help graphic

Parameter

Q305 Number in table?

Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303=1**, the control will write the data to the preset table.

If **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q405 New preset?

Coordinate in the measuring axis at which the control will set the calculated slot center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+9999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **0, 1**

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Help graphic

Parameter

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 408 SLOT CENTER PRESET ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q311=+25	;SLOT WIDTH ~
Q272=+1	;MEASURING AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+10	;NUMBER IN TABLE ~
Q405=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.21 Cycle 409 RIDGE CENTER PRESET

ISO programming

G409

Application

Touch probe cycle **409** finds the center of a ridge and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



Instead of Cycle **409 RIDGE CENTER PRESET**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

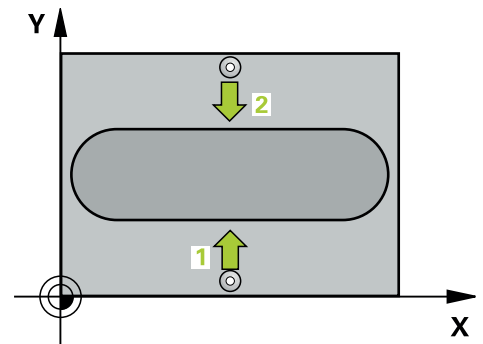
Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE", Page 149

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.
- 4 The control returns the touch probe to the clearance height.
- 5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Application", Page 165
- 6 Then the control saves the actual values in the Q parameters listed below.
- 7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE

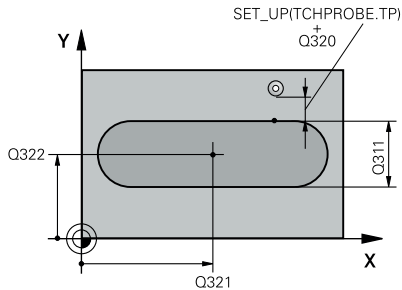
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

- ▶ Before the cycle definition, you must have programmed a tool call to define the touch probe axis.
-
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
 - The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the ridge in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the ridge in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q311 Ridge width?

Width of the ridge, regardless of its position in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis

Input: **1, 2**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

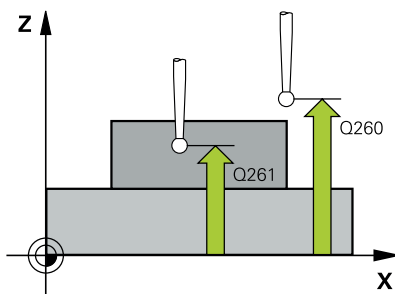
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**



Help graphic**Parameter****Q305 Number in table?**

Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303=1**, the control will write the data to the preset table.

If **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 166

Input: **0...99999**

Q405 New preset?

Coordinate in the measuring axis at which the control will set the calculated ridge center. Default setting = 0. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q303 Meas. value transfer (0,1)?

Define whether the calculated preset will be saved in the datum table or in the preset table:

0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system.

1: Write the calculated preset to the preset table.

Input: **0, 1**

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Q382 Probe TS axis: Coord. 1st axis?

Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Help graphic

Parameter

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

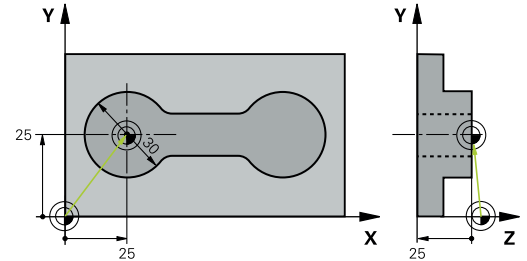
Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 409 RIDGE CENTER PRESET ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q311=+25	;RIDGE WIDTH ~
Q272=+1	;MEASURING AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q305=+10	;NUMBER IN TABLE ~
Q405=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.22 Example: Presetting at center of a circular segment and on top surface of workpiece

- **Q325** = Polar coordinate angle for touch point 1
- **Q247** = Stepping angle for calculating the touch points 2 to 4
- **Q305** = Write to row number 5 of the preset table
- **Q303** = Write the calculated preset to the preset table
- **Q381** = Also set the preset in the touch probe axis
- **Q365** = Move on circular path between measuring points

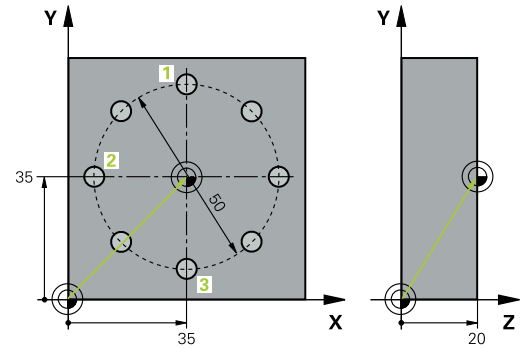


0 BEGIN PGM 413 MM	
1 TOOL CALL "TOUCH_PROBE" Z	
2 TCH PROBE 413 PRESET OUTS. CIRCLE ~	
Q321=+25	;CENTER IN 1ST AXIS ~
Q322=+25	;CENTER IN 2ND AXIS ~
Q262=+30	;NOMINAL DIAMETER ~
Q325=+90	;STARTING ANGLE ~
Q247=+45	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+5	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+10	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+25	;1ST CO. FOR TS AXIS ~
Q383=+25	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+0	;PRESET ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+0	;TYPE OF TRAVERSE
3 END PGM 413 MM	

5.23 Example: Presetting on top surface of workpiece and at center of a bolt hole circle

The control will write the measured bolt-hole circle center to the preset table so that it may be used at a later time.

- **Q291** = Polar coordinate angle for first hole center **1**
- **Q292** = Polar coordinate angle for second hole center **2**
- **Q293** = Polar coordinate angle for third hole center **3**
- **Q305** = Write center of bolt hole circle (X and Y) to row 1
- **Q303** = In the preset table **PRESET.PR**, save the calculated preset referenced to the machine-based coordinate system (REF system)



0 BEGIN PGM 416 MM	
1 TOOL CALL "TOUCH_PROBE" Z	
2 TCH PROBE 416 PRESET CIRCLE CENTER ~	
Q273=+35	;CENTER IN 1ST AXIS ~
Q274=+35	;CENTER IN 2ND AXIS ~
Q262=+50	;NOMINAL DIAMETER ~
Q291=+90	;ANGLE OF 1ST HOLE ~
Q292=+180	;ANGLE OF 2ND HOLE ~
Q293=+270	;ANGLE OF 3RD HOLE ~
Q261=+15	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q305=+1	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+7.5	;1ST CO. FOR TS AXIS ~
Q383=+7.5	;2ND CO. FOR TS AXIS ~
Q384=+20	;3RD CO. FOR TS AXIS ~
Q333=+0	;PRESET ~
Q320=+0	;SET-UP CLEARANCE.
3 CYCL DEF 247 PRESETTING ~	
Q339=+1	;PRESET NUMBER
4 END PGM 416 MM	

6

**Touch Probe
Cycles: Automatic
Workpiece
Inspection**

6.1 Fundamentals

Overview



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

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.







NOTICE







Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

The control offers twelve cycles for measuring workpieces automatically.

Soft key	Cycle	Page
	Cycle 0 REF. PLANE <ul style="list-style-type: none"> ■ Measuring a coordinate in a selectable axis 	240
	Cycle 1 POLAR PRESET <ul style="list-style-type: none"> ■ Measuring a point ■ Probing direction via angle 	242
	Cycle 420 MEASURE ANGLE <ul style="list-style-type: none"> ■ Measuring an angle in the working plane 	244
	Cycle 421 MEASURE HOLE <ul style="list-style-type: none"> ■ Measuring the position of a hole ■ Measuring the diameter of a hole ■ Nominal-to-actual value comparison, if applicable 	247
	Cycle 422 MEAS. CIRCLE OUTSIDE <ul style="list-style-type: none"> ■ Measuring the position of a circular stud ■ Measuring the diameter of a circular stud ■ Nominal-to-actual value comparison, if applicable 	253
	Cycle 423 MEAS. RECTAN. INSIDE <ul style="list-style-type: none"> ■ Measuring the position of a rectangular pocket ■ Measuring the length and width of a rectangular pocket ■ Nominal-to-actual value comparison, if applicable 	259

Soft key	Cycle	Page
	Cycle 424 MEAS. RECTAN. OUTS. <ul style="list-style-type: none"> ■ Measuring the position of a rectangular stud ■ Measuring the length and width of a rectangular stud ■ Nominal-to-actual value comparison, if applicable 	264
	Cycle 425 MEASURE INSIDE WIDTH <ul style="list-style-type: none"> ■ Measuring the position of a slot ■ Measuring the width of a slot ■ Nominal-to-actual value comparison, if applicable 	268
	Cycle 426 MEASURE RIDGE WIDTH <ul style="list-style-type: none"> ■ Measuring the position of a ridge ■ Measuring the width of a ridge ■ Nominal-to-actual value comparison, if applicable 	272
	Cycle 427 MEASURE COORDINATE <ul style="list-style-type: none"> ■ Measuring any coordinate in a selectable axis ■ Nominal-to-actual value comparison, if applicable 	276
	Cycle 430 MEAS. BOLT HOLE CIRC <ul style="list-style-type: none"> ■ Measuring the center point of a bolt hole circle ■ Measuring the diameter of a bolt hole circle ■ Nominal-to-actual value comparison, if applicable 	281
	Cycle 431 MEASURE PLANE <ul style="list-style-type: none"> ■ Finding the angle of a plane by measuring three points 	286

Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles **0** and **1**), you can have the control record the measurement results in a log. In the respective probing cycle you can define if the control is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log to a file, the control by default saves the data as an ASCII file. The control will save the file in the directory that also contains the associated NC program.

The unit of measurement of the main program can be seen in the header of the log file.



Use the HEIDENHAIN data transfer software TNCremo if you wish to output the measuring log over the data interface.

Example: Measuring log for touch probe cycle **421**:

Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005

Time: 6:55:04

Measuring program: TNC:\GEH35712\CHECK1.H

Type of dimension (0 = MM / 1 = INCH): 0

Nominal values:

Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000

Given limit values:

Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000

Minimum limit for center in minor axis:	64.9000
Maximum dimension for hole:	12.0450
Minimum dimension for hole:	12.0000

Actual values:

Center in reference axis:	50.0810
Center in minor axis:	64.9530
Diameter:	12.0259

Deviations:

Center in reference axis:	0.0810
Center in minor axis:	-0.0470
Diameter:	0.0259

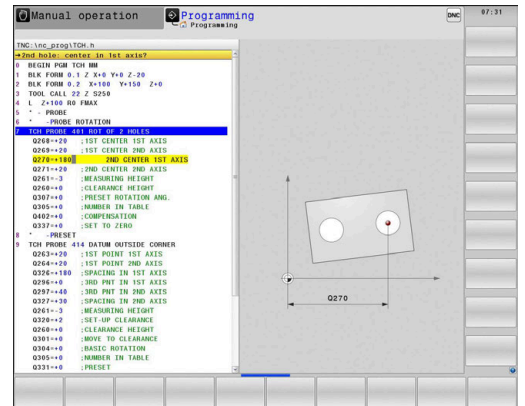
Further measuring results: Measuring height:	-5.0000
--	---------

End of measuring log

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. Deviations from the nominal values are saved in parameters **Q161** to **Q166**. Note the table of result parameters listed with every cycle description.

During cycle definition, the control also shows the result parameters for the respective cycle in a help graphic (see figure at right). The highlighted result parameter belongs to that input parameter.



Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters **Q180** to **Q182**.

Parameter value	Measuring status
Q180 = 1	Measurement results are within tolerance
Q181 = 1	Rework is required
Q182 = 1	Scrap

The control sets the rework or scrap marker as soon as one of the measuring values is out of tolerance. To determine which of the measuring results is out of tolerance, check the measuring log, or compare the respective measuring results (**Q150** to **Q160**) with their limit values.

In Cycle **427** the control assumes by default that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.

i 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 will be compensated due to the deviations from the nominal value (values in **Q16x**)
- the deviations from the nominal value (values in **Q16x**) are greater than the tool breakage tolerance.

Tool compensation

Requirements:

- Active tool table
- Tool monitoring must be switched on in the cycle: Set **Q330** unequal to 0 or enter a tool name. To enter the tool name, press the associated soft key. The control no longer displays the right single quotation mark.



- HEIDENHAIN recommends using this function only if the tool to be compensated for is the one that was used to machine the contour as well as if any necessary reworking will also be done with this tool.
- If you perform several compensation measurements, the control adds the respective measured deviation to the value stored in the tool table.

Milling 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: (applies only to Cycles **421, 422, 427**) If you reference a turning tool in parameter **Q330**, the appropriate values in row DZL and DXL, respectively, will be compensated. The control also monitors the breakage tolerance, which is defined in column LBREAK. Parameter **Q181** allows you to inquire in the NC program whether re-working is necessary (**Q181 = 1**: rework required).

If you want to compensate the values for an indexed tool with a tool name, program the following:

- **Q50** = "TOOL NAME"
- **FN 18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0**= **Q0** +0.2; add the index of the basic tool number
- In the cycle: **Q330 = Q0**; use the indexed tool

Tool breakage monitoring

Requirements:

- Active tool table
- Tool monitoring must be switched on in the cycle (set **Q330** unequal to 0)
- RBREAK must be greater than 0 (in the entered tool number in the table)

Further information: 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.

6.2 Cycle 0 REF. PLANE

ISO programming

G55

Application

The touch probe cycle measures any position on the workpiece in a selectable axis direction.



Instead of Cycle **0 REF. PLANE**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

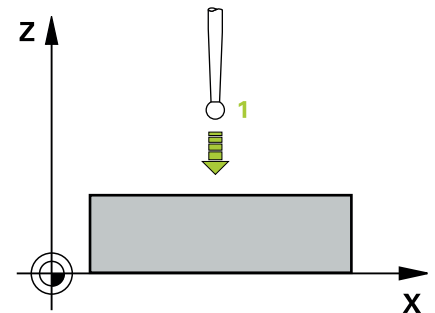
Related topics

- Cycle **1400 POSITION PROBING**

Further information: "Cycle 1400 POSITION PROBING", Page 135

Cycle run

- 1 In a 3D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
- 2 Next, the touch probe performs probing at the probing feed rate (**F** column). The probing direction must be defined in the cycle.
- 3 After the control has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. In addition, the control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**. For the values in these parameters the control does not account for the stylus length and radius.



Notes

NOTICE

Danger of collision!

The control moves the touch probe in a 3D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- ▶ Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

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

Cycle parameters

Help graphic	Parameter
	<p>Parameter number for result?</p> <p>Enter the number of the Q parameter to which you want to assign the coordinate..</p> <p>Input: 0...1999</p>
	<p>Probing axis/probing direction?</p> <p>Select the probing axis with the axis key or the alphabetic keyboard, entering the algebraic sign for the probing direction.</p> <p>Input: -, +</p>
	<p>Position value?</p> <p>Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe.</p> <p>Input: -999999999...+999999999</p>

Example

```
11 TCH PROBE 0.0 REF. PLANE Q9 Z+
```

```
12 TCH PROBE 0.1 X+99 Y+22 Z+2
```

6.3 Cycle 1 POLAR PRESET

ISO programming

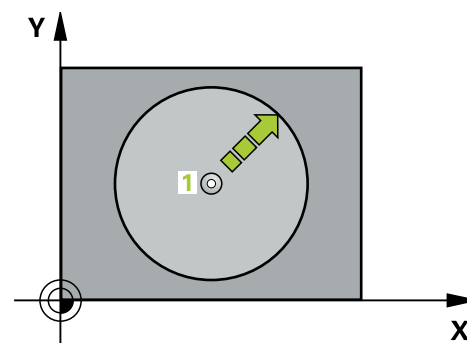
NC syntax is available only in Klartext programming.

Application

Touch probe cycle **1** measures any position on the workpiece in any probing direction.

Cycle sequence

- 1 In a 3D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
- 2 Next, the touch probe performs probing at the probing feed rate (**F** column). During probing, the control moves the touch probe simultaneously in two axes (depending on the probing angle). Use polar angles to define the probing direction in the cycle.
- 3 After the control has saved the position, the touch probe returns to the starting point. The control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**



Notes

NOTICE

Danger of collision!

The control moves the touch probe in a 3D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- ▶ Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The probing axis defined in the cycle specifies the probing plane:
 Probing axis X: X/Y plane
 Probing axis Y: Y/Z plane
 Probing axis Z: Z/X plane

Cycle parameters

Help graphic	Parameter
	<p>Probing axis?</p> <p>Enter the probing axis with the axis key or the alphabetic keyboard. Confirm with the ENT key.</p> <p>Input: X, Y, or Z</p>
	<p>Probing angle?</p> <p>Angle measured from the probing axis in which the touch probe will move.</p> <p>Input: -180...+180</p>
	<p>Position value?</p> <p>Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe.</p> <p>Input: -999999999...+999999999</p>

Example

```
11 TCH PROBE 1.0 POLAR PRESET
```

```
12 TCH PROBE 1.1 X ANGLE:+30
```

```
13 TCH PROBE 1.2 X+0 Y+10 Z+3
```

6.4 Cycle 420 MEASURE ANGLE

ISO programming

G420

Application

Touch probe cycle **420** measures the angle that any straight line on the workpiece forms with the main axis of the working plane.

i Instead of Cycle **420 MEASURE ANGLE**, HEIDENHAIN recommends using the more powerful Cycle **1410 PROBING ON EDGE**.

Related topics

- Cycle **1410 PROBING ON EDGE**

Further information: "Cycle 1410 PROBING ON EDGE", Page 71

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

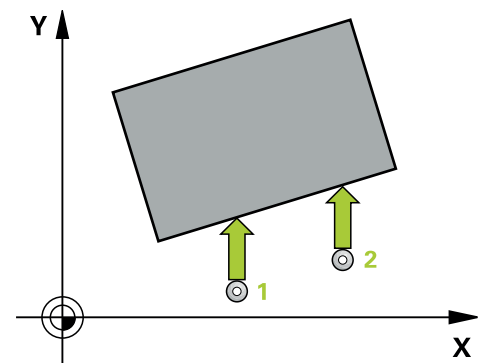
Q parameter number	Meaning
Q150	The measured angle is referenced to the main axis of the working plane.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:
 - If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
 - If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.
- The control will reset an active basic rotation at the beginning of the cycle.

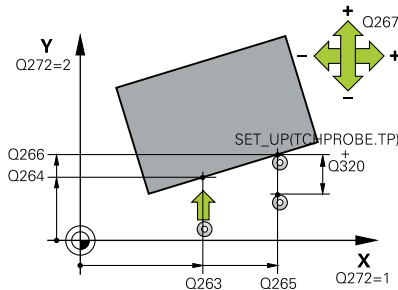
Note on programming

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



Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis
- 3: Touch probe axis = measuring axis

Input: **1, 2, 3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

- 1: Negative traverse direction
- +1: Positive traverse direction

Input: **-1, +1**

Q261 Measuring height in probe axis?

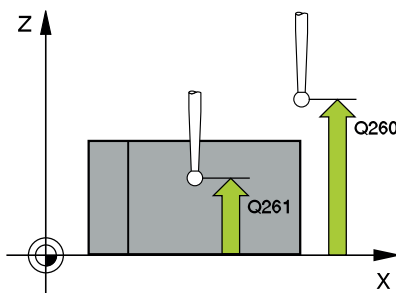
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of **Q320**, **SET_UP**, and the ball-tip radius, even when probing in the tool axis direction. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**



Help graphic

Parameter

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

Define whether the control will create a measuring log:

1: Create a measuring log: The control will save the **log file named TCHPR420.TXT** in the folder that also contains the associated NC program.

2: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with **NC Start**)

Input: **0, 1, 2**

Example

11 TCH PROBE 420 MEASURE ANGLE ~	
Q263=+10	;1ST POINT 1ST AXIS ~
Q264=+10	;1ST POINT 2ND AXIS ~
Q265=+15	;2ND PNT IN 1ST AXIS ~
Q266=+95	;2ND POINT 2ND AXIS ~
Q272=+1	;MEASURING AXIS ~
Q267=-1	;TRAVERSE DIRECTION ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+1	;MOVE TO CLEARANCE ~
Q281=+1	;MEASURING LOG

6.5 Cycle 421 MEASURE HOLE

ISO programming

G421

Application

Touch probe cycle **421** measures the center point and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



Instead of Cycle **421 MEASURE HOLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

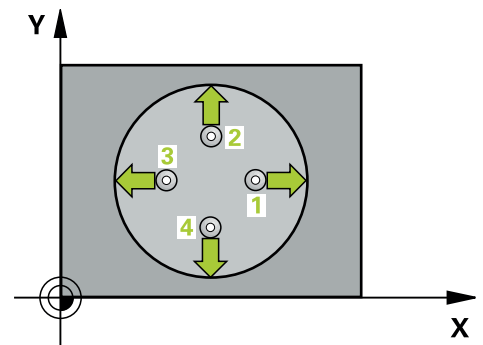
Related topics

- Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING", Page 139

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Notes

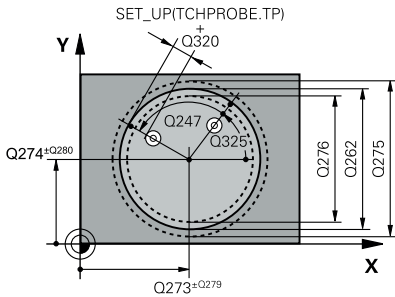
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The nominal diameter **Q262** must be between the minimum and maximum dimension (**Q276/Q275**).
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter **Q330** references a turning tool, the following applies:
 - Parameters **Q498** and **Q531** have to be defined
 - The information in parameters **Q498** and **Q531**, for example from Cycle **800**, has to match this information
 - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
 - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Center of the hole in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Center of the hole in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Enter the diameter of the hole.

Input: **0...99999.9999**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect.

Input: **-120...+120**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

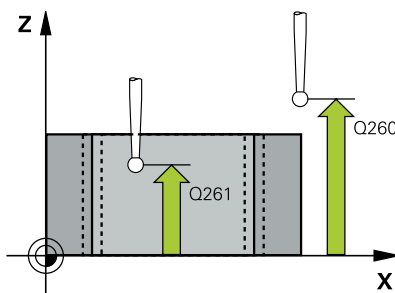
Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**



Help graphic

Parameter

Q275 Maximum limit of size for hole?

Maximum permissible diameter for the hole (circular pocket)

Input: **0...99999.9999**

Q276 Minimum limit of size?

Minimum permissible diameter for the hole (circular pocket)

Input: **0...99999.9999**

Q279 Tolerance for center 1st axis?

Permissible position deviation in the main axis of the working plane.

Input: **0...99999.9999**

Q280 Tolerance for center 2nd axis?

Permissible position deviation in the secondary axis of the working plane.

Input: **0...99999.9999**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR421.TXT** by default in the directory that also contains the associated NC program.

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via a soft key, you have the option of applying a tool directly from the tool table.

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Help graphic**Parameter****Q423 No. probe points in plane (4/3)?**

Define whether the control will use three or four touch points to measure the circle:

3: Use three measuring points

4: Use four measuring points (default setting)

Input: **3, 4**

Q365 Type of traverse? Line=0/arc=1

Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active.

0: Move in a straight line between machining operations

1: Move along a circular arc on the pitch circle diameter between machining operations

Input: **0, 1**

Q498 Reverse tool (0=no/1=yes)?

Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following:

1: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1

0: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 0

Input: **0, 1**

Q531 Angle of incidence?

Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**).

Input: **-180...+180**

Example

11 TCH PROBE 421 MEASURE HOLE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+15.25	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+1	;MOVE TO CLEARANCE ~
Q275=+15.34	;MAXIMUM LIMIT ~
Q276=+15.16	;MINIMUM LIMIT ~
Q279=+0.1	;TOLERANCE 1ST CENTER ~
Q280=+0.1	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE ~
Q498=+0	;REVERSE TOOL ~
Q531=+0	;ANGLE OF INCIDENCE

6.6 Cycle 422 MEAS. CIRCLE OUTSIDE

ISO programming

G422

Application

Touch probe cycle **422** measures the center point and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



Instead of Cycle **422 MEAS. CIRCLE OUTSIDE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

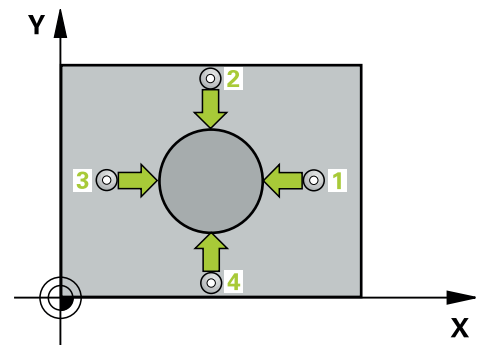
Related topics

- Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING", Page 139

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Notes

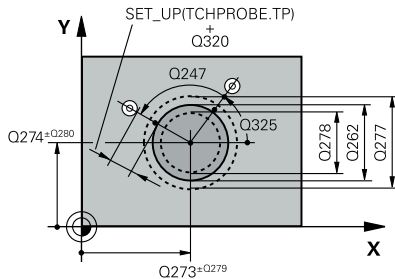
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter **Q330** references a turning tool, the following applies:
 - Parameters **Q498** and **Q531** have to be defined
 - The information in parameters **Q498** and **Q531**, for example from Cycle **800**, has to match this information
 - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
 - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Center of the stud in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Center of the stud in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Enter the diameter of the stud.

Input: **0...99999.9999**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two measuring points. The algebraic sign of the stepping angle determines the machining direction (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect.

Input: **-120...+120**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

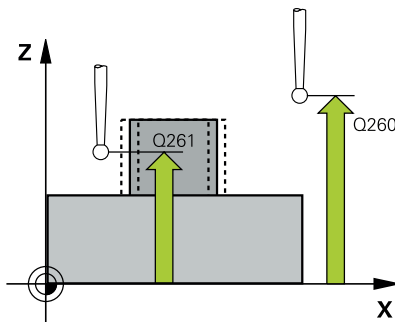
Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**



Help graphic

Parameter

Q277 Maximum limit of size for stud?

Maximum permissible diameter for the stud.

Input: **0...99999.9999**

Q278 Minimum limit of size for stud?

Minimum permissible diameter for the stud.

Input: **0...99999.9999**

Q279 Tolerance for center 1st axis?

Permissible position deviation in the main axis of the working plane.

Input: **0...99999.9999**

Q280 Tolerance for center 2nd axis?

Permissible position deviation in the secondary axis of the working plane.

Input: **0...99999.9999**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR422.TXT** in the folder that also contains the associated NC program.

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Tool number in tool table TOOL.T

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Q423 No. probe points in plane (4/3)?

Define whether the control will use three or four touch points to measure the circle:

3: Use three measuring points

4: Use four measuring points (default setting)

Input: **3, 4**

Help graphic**Parameter****Q365 Type of traverse? Line=0/arc=1**

Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active.

0: Move in a straight line between machining operations

1: Move along a circular arc on the pitch circle diameter between machining operations

Input: **0, 1**

Q498 Reverse tool (0=no/1=yes)?

Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following:

1: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1

0: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 0

Input: **0, 1**

Q531 Angle of incidence?

Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**).

Input: **-180...+180**

Example

11 TCH PROBE 422 MEAS. CIRCLE OUTSIDE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+90	;STARTING ANGLE ~
Q247=+30	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q277=+35.15	;MAXIMUM LIMIT ~
Q278=+34.9	;MINIMUM LIMIT ~
Q279=+0.05	;TOLERANCE 1ST CENTER ~
Q280=+0.05	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE ~
Q498=+0	;REVERSE TOOL ~
Q531=+0	;ANGLE OF INCIDENCE

6.7 Cycle 423 MEAS. RECTAN. INSIDE

ISO programming

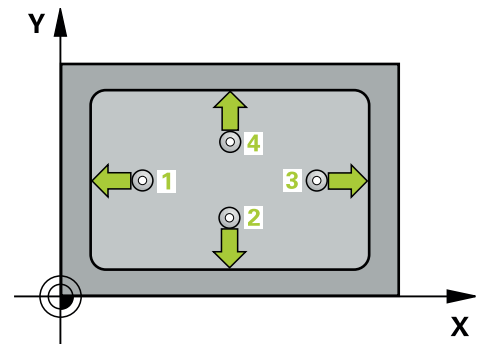
G423

Application

Touch probe cycle **423** finds the center, length, and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

Notes

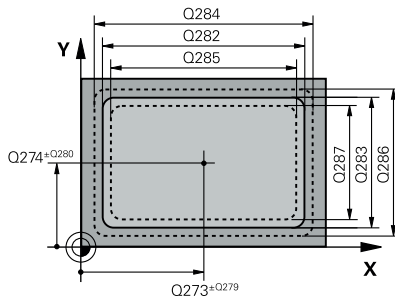
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Center of the pocket in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Center of the pocket in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q282 1st side length (nominal value)?

Pocket length, parallel to the main axis of the working plane

Input: **0...99999.9999**

Q283 2nd side length (nominal value)?

Pocket length, parallel to the secondary axis of the working plane

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q284 Max. size limit 1st side length?

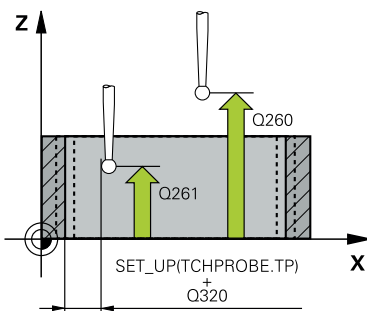
Maximum permissible length for the pocket

Input: **0...99999.9999**

Q285 Min. size limit 1st side length?

Minimum permissible length for the pocket

Input: **0...99999.9999**



Help graphic

Parameter

Q286 Max. size limit 2nd side length?

Maximum permissible width for the pocket

Input: **0...99999.9999**

Q287 Min. size limit 2nd side length?

Minimum permissible width for the pocket

Input: **0...99999.9999**

Q279 Tolerance for center 1st axis?

Permissible position deviation in the main axis of the working plane.

Input: **0...99999.9999**

Q280 Tolerance for center 2nd axis?

Permissible position deviation in the secondary axis of the working plane.

Input: **0...99999.9999**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log.

1: Create a measuring log: The control will save the **log file named TCHPR423.TXT** in the folder that also contains the associated NC program.

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Tool number in tool table TOOL.T

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Example

11 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q282=+80	;FIRST SIDE LENGTH ~
Q283=+60	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+1	;MOVE TO CLEARANCE ~
Q284=+0	;MAX. LIMIT 1ST SIDE ~
Q285=+0	;MIN. LIMIT 1ST SIDE ~
Q286=+0	;MAX. LIMIT 2ND SIDE ~
Q287=+0	;MIN. LIMIT 2ND SIDE ~
Q279=+0	;TOLERANCE 1ST CENTER ~
Q280=+0	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

6.8 Cycle 424 MEAS. RECTAN. OUTS.

ISO programming

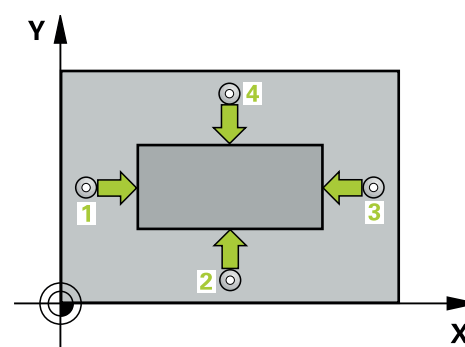
G424

Application

Touch probe cycle **424** finds the center, length, and width of a rectangular stud. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

Notes

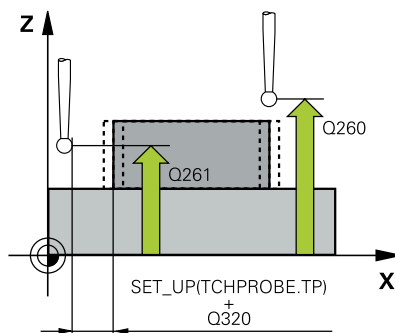
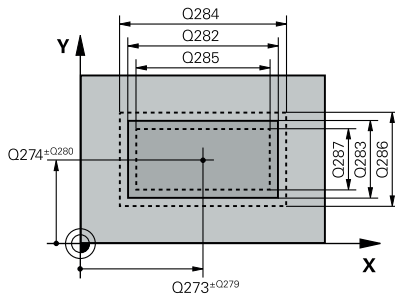
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Center of the stud in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Center of the stud in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q282 1st side length (nominal value)?

Length of stud parallel to the main axis of the working plane

Input: **0...99999.9999**

Q283 2nd side length (nominal value)?

Length of stud parallel to the secondary axis of the working plane

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q284 Max. size limit 1st side length?

Maximum permissible length for the stud

Input: **0...99999.9999**

Q285 Min. size limit 1st side length?

Minimum permissible length for the stud

Input: **0...99999.9999**

Help graphic

Parameter

Q286 Max. size limit 2nd side length?

Maximum permissible width for the stud

Input: **0...99999.9999**

Q287 Min. size limit 2nd side length?

Minimum permissible width for the stud

Input: **0...99999.9999**

Q279 Tolerance for center 1st axis?

Permissible position deviation in the main axis of the working plane.

Input: **0...99999.9999**

Q280 Tolerance for center 2nd axis?

Permissible position deviation in the secondary axis of the working plane.

Input: **0...99999.9999**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR424.TXT** in the folder that also contains the .h file

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via a soft key, you have the option of applying a tool directly from the tool table.

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Example

11 TCH PROBE 424 MEAS. RECTAN. OUTS. ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;2ND CENTER 2ND AXIS ~
Q282=+75	;FIRST SIDE LENGTH ~
Q283=+35	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q284=+75.1	;MAX. LIMIT 1ST SIDE ~
Q285=+74.9	;MIN. LIMIT 1ST SIDE ~
Q286=+35	;MAX. LIMIT 2ND SIDE ~
Q287=+34.95	;MIN. LIMIT 2ND SIDE ~
Q279=+0.1	;TOLERANCE 1ST CENTER ~
Q280=+0.1	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

6.9 Cycle 425 MEASURE INSIDE WIDTH

ISO programming

G425

Application

Touch probe cycle **425** measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation value in a Q parameter.



Instead of Cycle **425 MEASURE INSIDE WIDTH**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE", Page 149

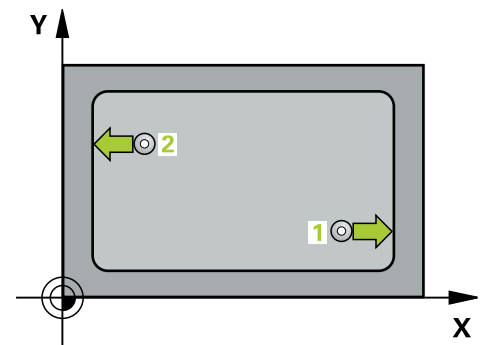
Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the positive direction of the programmed axis.
- 3 If you enter an offset for the second measurement, the control then moves the touch probe (if required, at clearance height) to the next touch point **2** and probes that point. If the nominal length is large, the control moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the control measures the width in the exact opposite direction.
- 4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

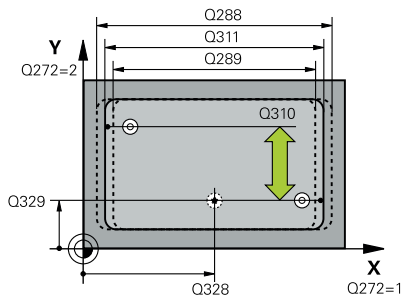


Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The nominal length **Q311** must be between the minimum and maximum dimension (**Q276/Q275**).

Cycle parameters

Help graphic



Parameter

Q328 Starting point in 1st axis?

Starting point for probing in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q329 Starting point in 2nd axis?

Starting point for probing in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q310 Offset for 2nd measurement (+/-)?

Distance by which the touch probe is offset before the second measurement. If you enter 0, the control does not offset the touch probe. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

Input: **1, 2**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q311 Nominal length?

Nominal value of the length to be measured

Input: **0...99999.9999**

Q288 Maximum limit of size?

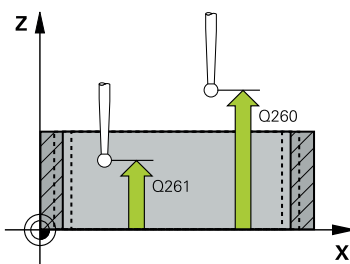
Maximum permissible length

Input: **0...99999.9999**

Q289 Minimum limit of size?

Minimum permissible length

Input: **0...99999.9999**



Help graphic

Parameter

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR425.TXT** in the folder that also contains the .h file

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via a soft key, you have the option of applying a tool directly from the tool table.

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Example

11 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

6.10 Cycle 426 MEASURE RIDGE WIDTH

ISO programming

G426

Application

Touch probe cycle **426** measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



Instead of Cycle **426 MEASURE RIDGE WIDTH**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE", Page 149

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 45
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the negative direction of the programmed axis.
- 3 Then the touch probe moves at clearance height to the next touch point and probes it.
- 4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

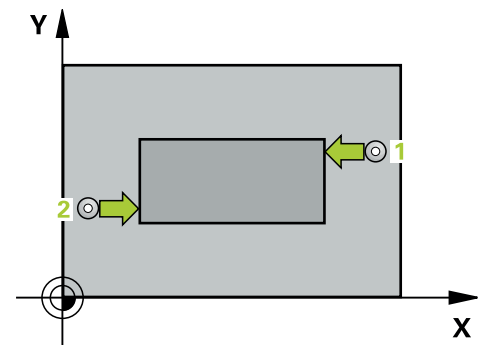
Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

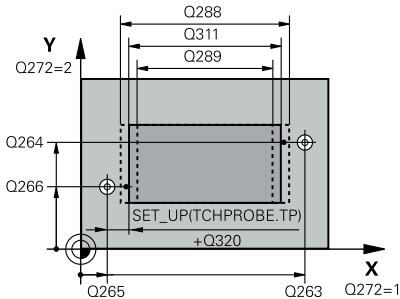
Note on programming

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



Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis

Input: **1, 2**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q311 Nominal length?

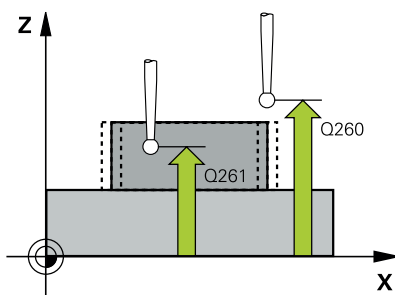
Nominal value of the length to be measured

Input: **0...99999.9999**

Q288 Maximum limit of size?

Maximum permissible length

Input: **0...99999.9999**



Help graphic

Parameter

Q289 Minimum limit of size?

Minimum permissible length

Input: **0...99999.9999**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR426.TXT** in the folder that also contains the associated NC program.

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via a soft key, you have the option of applying a tool directly from the tool table.

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Example

11 TCH PROBE 426 MEASURE RIDGE WIDTH ~	
Q263=+50	;1ST POINT 1ST AXIS ~
Q264=+25	;1ST POINT 2ND AXIS ~
Q265=+50	;2ND PNT IN 1ST AXIS ~
Q266=+85	;2ND PNT IN 2ND AXIS ~
Q272=+2	;MEASURING AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q311=+45	;NOMINAL LENGTH ~
Q288=+45	;MAXIMUM LIMIT ~
Q289=+44.95	;MINIMUM LIMIT ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

6.11 Cycle 427 MEASURE COORDINATE

ISO programming

G427

Application

Touch probe cycle **427** measures a coordinate in a selectable axis and saves the value in a Q parameter. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



Instead of Cycle **427 MEASURE COORDINATE**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

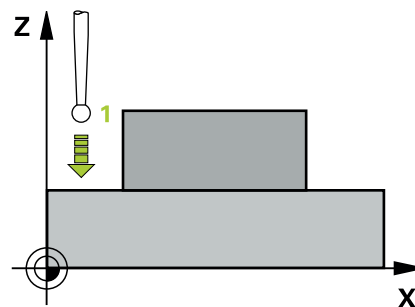
Further information: "Cycle 1400 POSITION PROBING", Page 135

Cycle run

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

Further information: "Positioning logic", Page 45

- 2 Then the control positions the touch probe to the specified touch point **1** in the working plane and measures the actual value in the selected axis.
- 3 Finally, the control returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:



Q parameter number	Meaning
--------------------	---------

Q160	Measured coordinate
------	---------------------

Notes

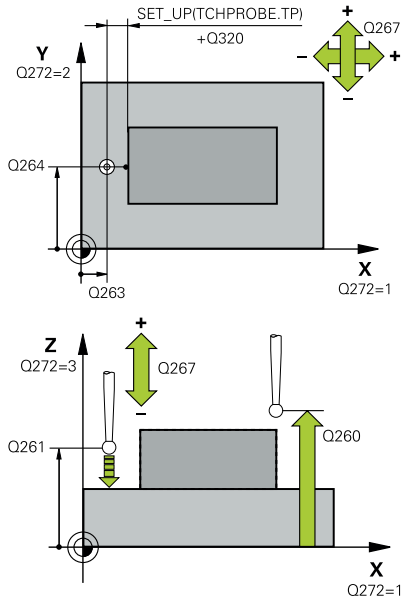
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).
- If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The measuring height **Q261** must be between the minimum and maximum dimension (**Q276/Q275**).
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter **Q330** references a turning tool, the following applies:
 - Parameters **Q498** and **Q531** have to be defined
 - The information in parameters **Q498** and **Q531**, for example from Cycle **800**, has to match this information
 - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
 - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis
- 3: Touch probe axis = measuring axis

Input: **1, 2, 3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

- 1: Negative traverse direction
- +1: Positive traverse direction

Input: **-1, +1**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic**Parameter****Q281 Measuring log (0/1/2)?**

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR427.TXT** in the folder that also contains the associated NC program.

2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q288 Maximum limit of size?

Maximum permissible value

Input: **-99999.9999...+99999.9999**

Q289 Minimum limit of size?

Minimum permissible value

Input: **-99999.9999...+99999.9999**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via a soft key, you have the option of applying a tool directly from the tool table.

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Help graphic

Parameter

Q498 Reverse tool (0=no/1=yes)?

Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following:

1: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498 = 1**

0: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example, Cycle **800** and parameter **Reverse the tool Q498 = 0**

Input: **0, 1**

Q531 Angle of incidence?

Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**).

Input: **-180...+180**

Example

11 TCH PROBE 427 MEASURE COORDINATE ~	
Q263=+35	;1ST POINT 1ST AXIS ~
Q264=+45	;1ST POINT 2ND AXIS ~
Q261=+5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q272=+3	;MEASURING AXIS ~
Q267=-1	;TRAVERSE DIRECTION ~
Q260=+20	;CLEARANCE HEIGHT ~
Q281=+1	;MEASURING LOG ~
Q288=+5.1	;MAXIMUM LIMIT ~
Q289=+4.95	;MINIMUM LIMIT ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL ~
Q498=+0	;REVERSE TOOL ~
Q531=+0	;ANGLE OF INCIDENCE

6.12 Cycle 430 MEAS. BOLT HOLE CIRC

ISO programming

G430

Application

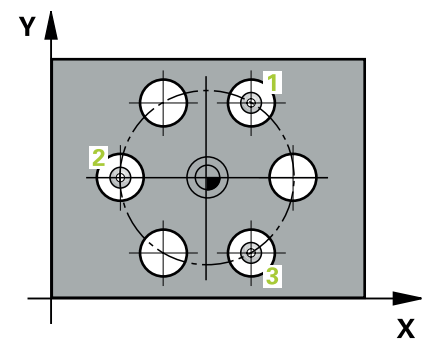
Touch probe cycle **430** finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle run

- 1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic

Further information: "Positioning logic", Page 45

- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
- 6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
- 7 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation of bolt circle diameter

Notes

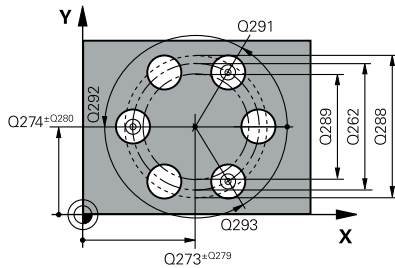
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **430** only monitors for tool breakage; there is no automatic tool compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Bolt hole circle center (nominal value) in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Bolt hole circle center (nominal value) in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Enter the diameter of the hole.

Input: **0...99999.9999**

Q291 Polar coord. angle of 1st hole?

Polar coordinate angle of the first hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q292 Polar coord. angle of 2nd hole?

Polar coordinate angle of the second hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q293 Polar coord. angle of 3rd hole?

Polar coordinate angle of the third hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q288 Maximum limit of size?

Maximum permissible diameter of bolt hole circle

Input: **0...99999.9999**

Q289 Minimum limit of size?

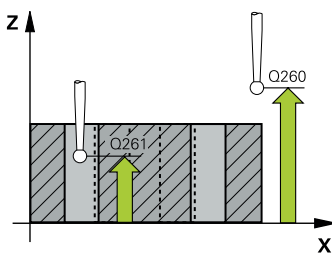
Minimum permissible diameter of bolt hole circle

Input: **0...99999.9999**

Q279 Tolerance for center 1st axis?

Permissible position deviation in the main axis of the working plane.

Input: **0...99999.9999**



Help graphic

Parameter

Q280 Tolerance for center 2nd axis?

Permissible position deviation in the secondary axis of the working plane.

Input: **0...99999.9999**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR430.TXT** in the folder that also contains the associated NC program

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via a soft key, you have the option of applying a tool directly from the tool table.

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 238

Example

11 TCH PROBE 430 MEAS. BOLT HOLE CIRC ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+80	;NOMINAL DIAMETER ~
Q291=+0	;ANGLE OF 1ST HOLE ~
Q292=+90	;ANGLE OF 2ND HOLE ~
Q293=+180	;ANGLE OF 3RD HOLE ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q288=+80.1	;MAXIMUM LIMIT ~
Q289=+79.9	;MINIMUM LIMIT ~
Q279=+0.15	;TOLERANCE 1ST CENTER ~
Q280=+0.15	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

6.13 Cycle 431 MEASURE PLANE

ISO programming

G431

Application

Touch probe cycle **431** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

i Instead of Cycle **431 MEASURE PLANE**, HEIDENHAIN recommends using the more powerful Cycle **1420 PROBING IN PLANE**.

Related topics

- Cycle **1420 PROBING IN PLANE**

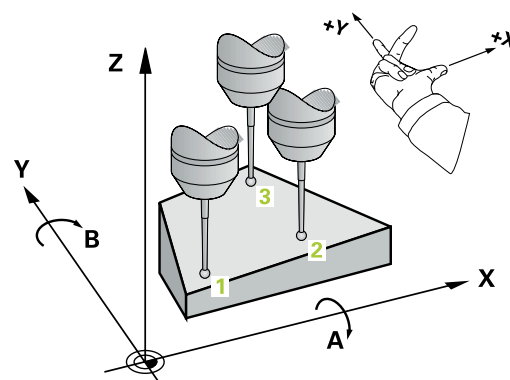
Further information: "Cycle 1420 PROBING IN PLANE", Page 64

Cycle run

- 1 The control positions the touch probe to the programmed touch point **1**, using positioning logic and measures the first plane point there. The control offsets the touch probe by the set-up clearance in the direction opposite to the direction of probing.

Further information: "Positioning logic", Page 45

- 2 The touch probe returns to the clearance height and then moves in the working plane to touch point **2** and measures the actual value of the second touch point in the plane.
- 3 The touch probe returns to the clearance height and then moves in the working plane to touch point **3** and measures the actual value of the third touch point in the plane.
- 4 Finally the control returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:



Q parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to Q175	Measured values in the touch probe axis (first to third measurement)

Notes

NOTICE

Risk of collision!

If you save the angle values in the preset table and then tilt the tool by programming **PLANE SPATIAL** with **SPA = 0; SPB = 0; SPC = 0**, there are multiple solutions in which the tilting axes are at 0. There is a risk of collision!

► Make sure to program **SYM (SEQ) +** or **SYM (SEQ) -**

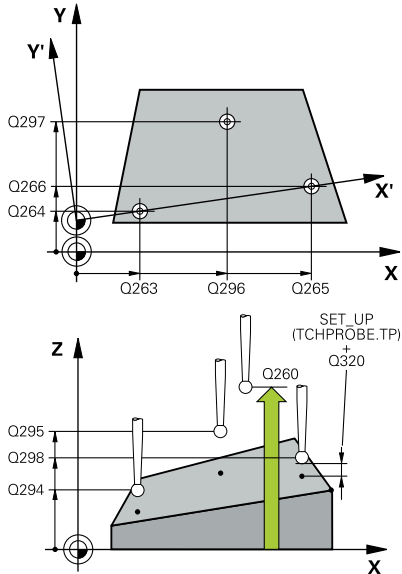
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three measuring points are not positioned on a straight line.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The spatial angles that are needed for the **Tilt working plane** function are saved in parameters **Q170** to **Q172**. With the first two measuring points, you also specify the direction of the main axis when tilting the working plane.
- The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q294 1st measuring point in 3rd axis?

Coordinate of the first touch point in the touch probe axis. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q295 2nd measuring point in 3rd axis?

Coordinate of the second touch point in the touch probe axis. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q296 3rd measuring point in 1st axis?

Coordinate of the third touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q297 3rd measuring point in 2nd axis?

Coordinate of the third touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q298 3rd measuring point in 3rd axis?

Coordinate of the third touch point in the touch probe axis. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Help graphic

Parameter

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the folder that also contains the associated NC program

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Example

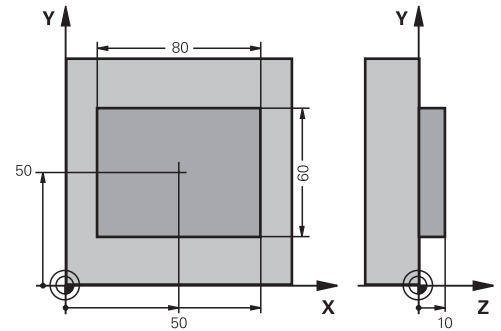
11 TCH PROBE 431 MEASURE PLANE ~	
Q263=+20	;1ST POINT 1ST AXIS ~
Q264=+20	;1ST POINT 2ND AXIS ~
Q294=-10	;1ST POINT 3RD AXIS ~
Q265=+50	;2ND PNT IN 1ST AXIS ~
Q266=+80	;2ND PNT IN 2ND AXIS ~
Q295=+0	;2ND PNT IN 3RD AXIS ~
Q296=+90	;3RD PNT IN 1ST AXIS ~
Q297=+35	;THIRD POINT 2ND AXIS ~
Q298=+12	;3RD PNT IN 3RD AXIS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+5	;CLEARANCE HEIGHT ~
Q281=+1	;MEASURING LOG

6.14 Programming examples

Example: Measuring and reworking a rectangular stud

Program sequence

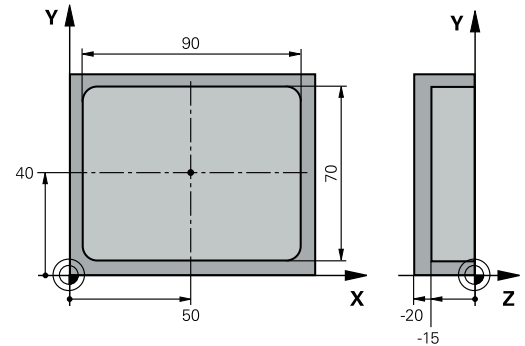
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



0	BEGIN PGM TOUCHPROBE MM	
1	TOOL CALL 5 Z S6000	; Tool call: roughing
2	Q1 = 81	; Rectangle length in X (roughing dimension)
3	Q2 = 61	; Rectangle length in Y (roughing dimension)
4	L Z+100 R0 FMAX M3	; Retract the tool
5	CALL LBL 1	; Call the subprogram for machining
6	L Z+100 R0 FMAX	; Retract the tool
7	TOOL CALL 600 Z	; Call the touch probe
8	TCH PROBE 424 MEAS. RECTAN. OUTS. ~	
	Q273=+50 ;CENTER IN 1ST AXIS ~	
	Q274=+50 ;CENTER IN 2ND AXIS ~	
	Q282=+80 ;FIRST SIDE LENGTH ~	
	Q283=+60 ;2ND SIDE LENGTH ~	
	Q261=-5 ;MEASURING HEIGHT ~	
	Q320=+0 ;SET-UP CLEARANCE ~	
	Q260=+30 ;CLEARANCE HEIGHT ~	
	Q301=+0 ;MOVE TO CLEARANCE ~	
	Q284=+0 ;MAX. LIMIT 1ST SIDE ~	
	Q285=+0 ;MIN. LIMIT 1ST SIDE ~	
	Q286=+0 ;MAX. LIMIT 2ND SIDE ~	
	Q287=+0 ;MIN. LIMIT 2ND SIDE ~	
	Q279=+0 ;TOLERANCE 1ST CENTER ~	
	Q280=+0 ;TOLERANCE 2ND CENTER ~	
	Q281=+0 ;MEASURING LOG ~	
	Q309=+0 ;PGM STOP TOLERANCE ~	
	Q330=+0 ;TOOL	
9	Q1 = Q1 - Q164	; Calculate the length in X based on the measured deviation
10	Q2 = Q2 - Q165	; Calculate the length in Y based on the measured deviation
11	L Z+100 R0 FMAX	; Retract the touch probe
12	TOOL CALL 25 Z S8000	; Tool call: finishing
13	L Z+100 R0 FMAX M3	; Retract the tool

14 CALL LBL 1	; Call the subprogram for machining
15 L Z+100 R0 FMAX	
16 M30	; End of program
17 LBL 1	; Subprogram with rectangular stud machining cycle
18 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+Q1	;FIRST SIDE LENGTH ~
Q424=+82	;WORKPC. BLANK SIDE 1 ~
Q219=+Q2	;2ND SIDE LENGTH ~
Q425=+62	;WORKPC. BLANK SIDE 2 ~
Q220=+0	;RADIUS / CHAMFER ~
Q368=+0.1	;ALLOWANCE FOR SIDE ~
Q224=+0	;ANGLE OF ROTATION ~
Q367=+0	;STUD POSITION ~
Q207=+500	;FEED RATE MILLING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q201=-10	;DEPTH ~
Q202=+5	;PLUNGING DEPTH ~
Q206=+3000	;FEED RATE FOR PLNGNG ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+10	;SURFACE COORDINATE ~
Q204=+20	;2ND SET-UP CLEARANCE ~
Q370=+1	;TOOL PATH OVERLAP ~
Q437=+0	;APPROACH POSITION ~
Q215=+0	;MACHINING OPERATION ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q338=+20	;INFEEED FOR FINISHING ~
Q385=+500	;FINISHING FEED RATE
19 L X+50 Y+50 R0 FMAX M99	; Cycle call
20 LBL 0	; End of subprogram
21 END PGM TOUCHPROBE MM	

Example: Probing a rectangular pocket and recording the results



0 BEGIN PGM TOUCHPROBE_2 MM	
1 TOOL CALL 600 Z	; Tool call: touch probe
2 L Z+100 R0 FMAX	; Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+40	;CENTER IN 2ND AXIS ~
Q282=+90	;FIRST SIDE LENGTH ~
Q283=+70	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q284=+90.15	;MAX. LIMIT 1ST SIDE ~
Q285=+89.95	;MIN. LIMIT 1ST SIDE ~
Q286=+70.1	;MAX. LIMIT 2ND SIDE ~
Q287=+69.9	;MIN. LIMIT 2ND SIDE ~
Q279=+0.15	;TOLERANCE 1ST CENTER ~
Q280=+0.1	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL
4 L Z+100 R0 FMAX	; Retract the tool
5 M30	; End of program
6 END PGM TOUCHPROBE_2 MM	

7

**Touch Probe Cycles:
Special Functions**

7.1 Fundamentals

Overview



The control must be specially prepared by the machine manufacturer for the use of a 3D touch probe.
HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.






NOTICE

Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

The control provides cycles for the following special purposes:

Soft key	Cycle	Page
	Cycle 3 MEASURING <ul style="list-style-type: none"> ■ Touch probe cycle for defining OEM cycles 	295
	Cycle 4 MEASURING IN 3-D <ul style="list-style-type: none"> ■ Measuring any position 	298
	Cycle 444 PROBING IN 3-D <ul style="list-style-type: none"> ■ Measuring any position ■ Determining the deviation from the nominal coordinates 	300
	Cycle 441 FAST PROBING <ul style="list-style-type: none"> ■ Touch probe cycle for defining various touch probe parameters 	306
	Cycle 1493 EXTRUSION PROBING <ul style="list-style-type: none"> ■ Touch probe cycle for defining an extrusion ■ Extrusion direction, length, and number of extrusion points can be programmed 	309

7.2 Cycle 3 MEASURING

ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **3** measures any position on the workpiece in a selectable probing direction. Unlike other touch probe cycles, Cycle **3** enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by a definable value **MB** after determining the measured value.

Cycle sequence

- 1 The touch probe moves from the current position at the specified feed rate in the defined probing direction. Use polar angles to define the probing direction in the cycle.
- 2 After the control has saved the position, the touch probe stops. The control saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The control does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.
- 3 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Notes



The exact behavior of touch probe cycle **3** is defined by your machine manufacturer or a software manufacturer who uses it within specific touch probe cycles.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other touch probe cycles, do not apply in touch probe cycle **3**.
- Remember that the control always writes to four successive Q parameters.
- If the control was not able to determine a valid touch point, the NC program is run without an error message. In this case the control assigns the value -1 to the fourth result parameter so that you can deal with the error yourself.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.



The **FN 17: SYSWRITE ID990 NR6** function allows setting whether the cycle runs through the probe input X12 or X13.

Cycle parameters

Help graphic

Parameter

Parameter number for result?

Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters.

Input: **0...1999**

Probing axis?

Enter the axis in whose direction the touch probe will move and confirm with the **ENT** key.

Input: **X, Y, or Z**

Probing angle?

This angle defines the probing direction. The angle refers to the probe axis. Confirm with the **ENT** key.

Input: **-180...+180**

Maximum measuring range?

Enter the maximum distance from the starting point by which the touch probe will move. Confirm with **ENT**.

Input: **0...999999999**

Feed rate measurement

Enter the measuring feed rate in mm/min.

Input: **0...3000**

Maximum retraction distance?

Traverse path in the direction opposite to the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision.

Input: **0...999999999**

Reference system? (0=ACT/1=REF)

Define whether the probing direction and measurement result will be referenced to the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**):

0: Perform the probing operation in the current system and save the measurement result in the **ACT** system

1: Perform the probing operation in the machine-based REF system. Save the measurement result in the REF system.

Input: **0, 1**

Help graphic**Parameter****Error mode? (0=OFF/1=ON)**

Define whether the control will issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the control saves the value **-1** in the 4th result parameter and continues the cycle:

0: Issue error message

1: Do not issue error message

Input: **0, 1**

Example

```
11 TCH PROBE 3.0 MEASURING
```

```
12 TCH PROBE 3.1 Q1
```

```
13 TCH PROBE 3.2 X ANGLE:+15
```

```
14 TCH PROBE 3.3 ABST+10 F100 MB1 REFERENCE SYSTEM:0
```

```
15 TCH PROBE 3.4 ERRORMODE1
```

7.3 Cycle 4 MEASURING IN 3-D

ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **4** measures any position on the workpiece in the probing direction defined by a vector. Unlike other touch probe cycles, Cycle **4** enables you to enter the probing distance and probing feed rate directly. You can also define the distance by which the touch probe retracts after acquiring the probed value.

Cycle **4** is an auxiliary cycle that can be used for probing with any touch probe (TS or TT). The control does not provide a cycle for calibrating the TS touch probe in any probing direction.

Cycle sequence

- 1 The control moves the touch probe from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).
- 2 After the control has saved the position, the control stops the probe movement. The control saves the X, Y, Z coordinates of the probing position in three successive Q parameters. You define the number of the first parameter in the cycle. If you are using a TS touch probe, the probe result is corrected by the calibrated center offset.
- 3 Finally, the control retracts the touch probe in the direction opposite to the direction of probing. You define the traverse distance in parameter **MB**—the touch probe is moved to a point no farther than the starting point.



Ensure during pre-positioning that the control moves the probe-tip center without compensation to the defined position.

Notes

NOTICE

Danger of collision!

If the control was not able to determine a valid touch point, the 4th result parameter will have the value **-1**. The control does **not** interrupt the program run! There is a danger of collision!

- ▶ Make sure that all touch points can be reached.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.
- Remember that the control always writes to four successive Q parameters.

Cycle parameters

Help graphic	Parameter
	<p>Parameter number for result?</p> <p>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.</p> <p>Input: 0...1999</p>
	<p>Relative measuring path in X?</p> <p>X component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: -999999999...+999999999</p>
	<p>Relative measuring path in Y?</p> <p>Y component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: -999999999...+999999999</p>
	<p>Relative measuring path in Z?</p> <p>Z component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: -999999999...+999999999</p>
	<p>Maximum measuring range?</p> <p>Enter the maximum distance from the starting point by which the touch probe will move along the direction vector.</p> <p>Input: -999999999...+999999999</p>
	<p>Feed rate measurement</p> <p>Enter the measuring feed rate in mm/min.</p> <p>Input: 0...3000</p>
	<p>Maximum retraction distance?</p> <p>Traverse path in the direction opposite the probing direction, after the stylus was deflected.</p> <p>Input: 0...999999999</p>
	<p>Reference system? (0=ACT/1=REF)</p> <p>Define whether the result of probing will be saved in the input coordinate system (ACT), or with respect to the machine coordinate system (REF):</p> <p>0: Save the measurement result in the ACT system</p> <p>1: Save the measurement result in the REF system</p> <p>Input: 0, 1</p>

Example

11 TCH PROBE 4.0 MEASURING IN 3-D

12 TCH PROBE 4.1 Q1

13 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1

14 TCH PROBE 4.3 ABST+45 F100 MB50 REFERENCE SYSTEM:0

7.4 Cycle 444 PROBING IN 3-D

ISO programming

G444

Application

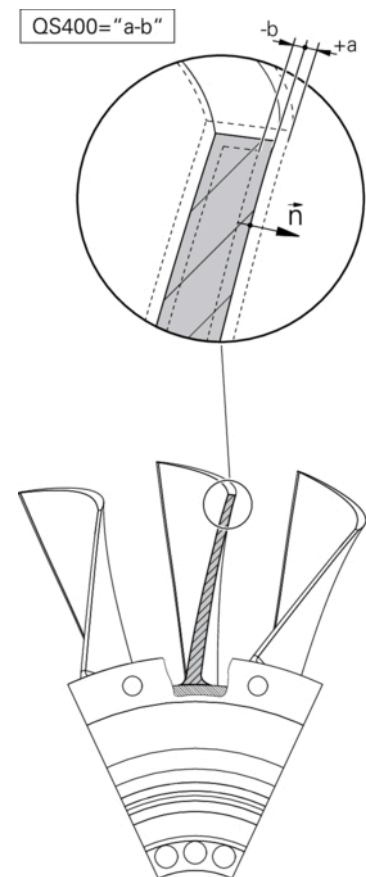


Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

Cycle **444** checks one specific point on the surface of a part. This cycle is used, for example, to measure free-form surfaces of moldmaking parts. It can be determined whether a point on the surface of the part lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.

Cycle **444** probes any point in three dimensions and determines the deviation from a nominal coordinate. A normal vector, defined in parameters **Q581**, **Q582**, and **Q583**, is used for this purpose. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range **QS400** defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the control outputs a log and the deviations are stored in the Q parameters listed below.

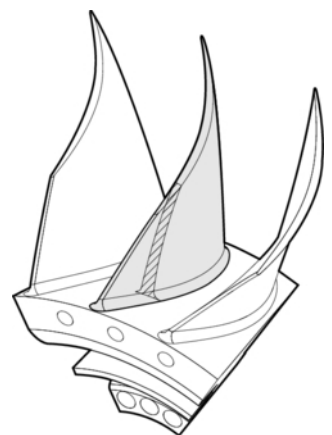


Cycle run

- 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.

Further information: "Executing touch probe cycles", Page 44

- 2 The touch probe then approaches the nominal coordinate. The probing distance is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.
- 3 After the control has saved the position, the touch probe is retracted and stopped. The control saves the measured coordinates of the contact point in Q parameters.
- 4 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.



Result parameters

The control stores the probing results in the following parameters:

Q parameter number	Meaning
Q151	Measured position in main axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in main axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3D deviation <ul style="list-style-type: none"> ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: <ul style="list-style-type: none"> ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

Log function

Once probing has finished, the control generates a log in HTML format. The log includes the results from the main, secondary, and tool axes as well as the 3D error. 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 **FN 16**).

The log contains the following data on the main, secondary, and tool axes:

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- If a tolerance **QS400** was defined: Upper and lower dimensions are output, as well as the determined deviation along the normal vector
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- In order to obtain exact results from the touch probe being used, you need to perform 3D calibration before executing Cycle **444**. 3D calibration requires **3D-ToolComp** (software option 92).
- Cycle **444** generates a measuring log in HTML format.
- An error message is output if Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, or Cycle **26 AXIS-SPECIFIC SCALING** is active before Cycle **444** is run.
- For probing, an active TCPM will be taken into account. While the TCPM is active, probing of positions is possible even if the position resulting from the **Tilt working plane** function is inconsistent with the current position of the rotary axes.
- If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.
- Cycle **444** references all coordinates to the input system.
- The control writes the measured values to return parameters.
Further information: "Application", Page 300
- The workpiece status good/rework/scrap is set via Q parameter **Q183**, independent of parameter **Q309**.
Further information: "Application", Page 300

Note regarding machine parameters

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

Cycle parameters

Help graphic	Parameter
	<p>Q263 1st measuring point in 1st axis?</p> <p>Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q264 1st measuring point in 2nd axis?</p> <p>Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q294 1st measuring point in 3rd axis?</p> <p>Coordinate of the first touch point in the touch probe axis. This value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q581 Surface-normal in ref. axis?</p> <p>Enter here the surface normal in the direction of the main axis. The surface normal of a point is normally output by a CAD/CAM system.</p> <p>Input: -10...+10</p>
	<p>Q582 Surface-normal in minor axis?</p> <p>Enter here the surface normal in the direction of the secondary axis. The surface normal of a point is normally output by a CAD/CAM system.</p> <p>Input: -10...+10</p>
	<p>Q583 Surface-normal in tool axis?</p> <p>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.</p> <p>Input: -10...+10</p>
	<p>Q320 Set-up clearance?</p> <p>Additional distance between touch point and ball tip. Q320 is active in addition to the SET_UP column in the touch probe table. This value has an incremental effect.</p> <p>Input: 0...99999.9999 or PREDEF</p>
	<p>Q260 Clearance height?</p> <p>Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999 or PREDEF</p>

Help graphic

Parameter

QS400 Tolerance value?

Specify a tolerance band that will be monitored by the cycle. The tolerance defines the deviation permitted along the surface normal. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. (The surface normal is defined by **Q581** to **Q583**, and the nominal coordinate is defined by **Q263**, **Q264**, and **Q294**.) The tolerance value is distributed over the axes, depending on the normal vector (see examples).

Examples

- **QS400 = "0.4-0.1"** means: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate -0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate -0.1"
- **QS400 = "0.4"** means: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate".
- **QS400 = "-0.1"** means: Upper dimension = nominal coordinate; lower dimension = nominal coordinate -0.1. The following tolerance band thus results for the cycle: "nominal coordinate" to "nominal coordinate -0.1".
- **QS400 = " "** means: No tolerance band.
- **QS400 = "0"** means: No tolerance band.
- **QS400 = "0.1+0.1"** means: No tolerance band.

Input: Max. **255** characters

Q309 Reaction to tolerance error?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run when tolerance is exceeded; do not output an error message

1: Interrupt program run when tolerance is exceeded and output an error message

2: If the value of the measured actual coordinate along the surface normal vector is less than the nominal coordinate, the control displays a message and interrupts the NC program run. However, there will be no error message if the value of the measured actual coordinate is greater than the nominal coordinate.

Input: **0, 1, 2**

Example

11 TCH PROBE 444 PROBING IN 3-D ~	
Q263=+0	;1ST POINT 1ST AXIS ~
Q264=+0	;1ST POINT 2ND AXIS ~
Q294=+0	;1ST POINT 3RD AXIS ~
Q581=+1	;NORMAL IN REF. AXIS ~
Q582=+0	;NORMAL IN MINOR AXIS ~
Q583=+0	;NORMAL IN TOOL AXIS ~
Q320=+0	;SAFETY CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
QS400="1-1"	;TOLERANCE ~
Q309=+0	;ERROR REACTION

7.5 Cycle 441 FAST PROBING

ISO programming

G441

Application

You can use touch probe cycle **441** to globally specify various touch probe parameters (e.g., the positioning feed rate) for all subsequently used touch probe cycles.



In this cycle, no machine movements will be performed.

Program interruption Q400=1

Parameter **Q400 INTERRUPTION** allows interrupting the cycle run and displaying the obtained results.

Program interruption by **Q400** is effective in the following touch probe cycles:

- Touch probe cycles for checking the workpiece: **421 to 427, 430** and **431**
- Cycle **444 PROBING IN 3-D**
- Touch probe cycles for measuring the kinematics: **45x**
- Touch probe cycles for calibrating: **46x**
- Touch probe cycles **14xx**

Cycles 421 to 427, 430 and 431:

The control displays the results obtained during a program interruption in a **FN 16** monitor output.

Cycles 444, 45x, 46x, 14xx:

The control automatically shows the results obtained during a program interruption in an HTML log in the path:
TNC:\TCHPRlast.html.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- **END PGM, M2, M30** reset the global settings of Cycle **441**.
- Cycle parameter **Q399** depends on your machine configuration. Your machine manufacturer is responsible for the setting of whether the touch probe can be oriented through an NC program.
- Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397=1**.
- If **Q371** is unequal to **0** and the stylus does not move in cycles **14xx**, the control will terminate the cycle. The control returns the touch probe to the clearance height and saves the workpiece status **3** in Q parameter **Q183**. The NC program continues.
Workpiece status **3**: Stylus does not move

Note regarding machine parameters

- The machine parameter **maxTouchFeed** (no. 122602) allows the machine manufacturer to limit the feed rate. You define the maximum absolute feed rate in this machine parameter.

Cycle parameters

Help graphic	Parameter
	<p>Q396 Positioning feed rate?</p> <p>Define the feed rate at which the touch probe will be moved to the specified positions.</p> <p>Input: 0...99999.999</p>
	<p>Q397 Pre-pos. at machine's rapid?</p> <p>Define whether the control, when repositioning the touch probe, traverses at FMAX feed rate (machine's rapid traverse):</p> <p>0: Pre-position at the feed rate from Q396</p> <p>1: Pre-position at the machine's rapid traverse FMAX</p> <p>Input: 0, 1</p>
	<p>Q399 Angle tracking (0/1)?</p> <p>Define whether the control will orient the touch probe before every probing operation:</p> <p>0: Do not orient the spindle</p> <p>1: Orient the spindle before every probing operation (increased accuracy)?</p> <p>Input: 0, 1</p>
	<p>Q400 Automatic interruption?</p> <p>Define whether the control will interrupt program run and output the measurement results on the screen following a touch probe cycle:</p> <p>0: Do not interrupt program run even if, in the specific touch probe cycle, the output of measurement results on the screen is selected</p> <p>1: Interrupt program run and output measurement results on the screen. You can then resume the NC program run with NC Start.</p> <p>Input: 0, 1</p> <p>Further information: "Program interruption Q400=1", Page 306</p>
	<p>Q371 Touch point not reached?</p> <p>Define how the control behaves when the stylus does not move within the DIST value of the touch probe table.</p> <p>0: The control interrupts the NC program with an error message saying that the touch point cannot be reached. This is standard behavior.</p> <p>1: The control displays a warning and terminates the probing cycle. The NC program continues. Is effective only in the 14xx cycles.</p> <p>2: The control displays no warning and terminates the probing cycle. The NC program continues. Is effective only in the 14xx cycles.</p> <p>Input: 0, 1, 2</p>

Example

11 TCH PROBE 441 FAST PROBING ~	
Q396=+3000	;POSITIONING FEEDRATE ~
Q397=+0	;SELECT FEED RATE ~
Q399=+1	;ANGLE TRACKING ~
Q400=+1	;INTERRUPTION ~
Q371=+0	;TOUCH POINT REACTION

7.6 Cycle 1493 EXTRUSION PROBING

ISO programming

G1493

Application

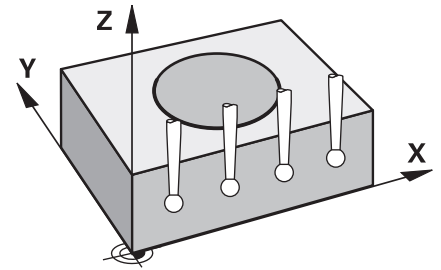
Cycle **1493** allows you to repeat the touch points of specific touch probe cycles along a straight line. In the cycle, you define the direction and the length of the extrusion, as well as the number of extrusion points.

The repetitions allow you, for example, to perform multiple measurements at different heights and to determine deviations based on the deflection of the tool. You can also use the extrusion to increase the accuracy during probing. Multiple measuring points help you ascertain contamination on the workpiece or rough surfaces.

In order to activate the repetition of specific touch points, you need to define Cycle **1493** before the probing cycle. Depending on the definition, this cycle will remain active for only the next cycle or for the entire NC program. The control interprets the extrusion in the input coordinate system **I-CS**.

The following cycles are capable of performing extrusions:

- **PROBING IN PLANE** (Cycle **1420**, ISO: **G1420**), see Page 64
- **PROBING ON EDGE** (Cycle **1410**, ISO: **G1410**), see Page 71
- **PROBING TWO CIRCLES** (Cycle **1411**, ISO: **G1411**), see Page 79
- **INCLINED EDGE PROBING** (Cycle **1412**, ISO: **G1412**), see Page 87
- **INTERSECTION PROBING** (Cycle **1416**, ISO: **G1416**), see Page 95
- **POSITION PROBING** (Cycle **1400**, ISO: **G1400**), see Page 135
- **CIRCLE PROBING** (Cycle **1401**, ISO: **G1401**), see Page 139
- **PROBE SLOT/RIDGE** (Cycle **1404**, ISO: **G1404**), see Page 149
- **PROBE POSITION OF UNDERCUT** (Cycle **1430**, ISO: **G1430**), see Page 153
- **PROBE SLOT/RIDGE UNDERCUT** (Cycle **1434**, ISO: **G1434**), see Page 159



Result parameter Q

The control saves the results of the touch probe cycle in the following Q parameters:

Q parameter number	Meaning
Q970	Maximum deviation from the ideal line of touch point 1
Q971	Maximum deviation from the ideal line of touch point 2
Q972	Maximum deviation from the ideal line of touch point 3
Q973	Maximum deviation of diameter 1
Q974	Maximum deviation of diameter 2

Result parameter QS

The control saves the individual results of all measuring points of an extrusion in the QS parameters **QS97x**. The result is ten characters long. The results are separated from each other by a space.

Example: **QS970 = 0.12345678 -1.1234567 -2.1234567 -3.1234567**

QS parameter number	Meaning
QS970	Results of touch point 1 of an extrusion
QS971	Results of touch point 2 of an extrusion
QS972	Results of touch point 3 of an extrusion
QS973	Results of diameter 1 of an extrusion
QS974	Results of diameter 2 of an extrusion

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

A touch probe cycle produces the following results within QS parameter **QS970**:

QS970 = 0.12345678 -1.1234567

The example below shows how to convert the results produced into numerical values.

11 QSO = SUBSTR (SRC_QS970 BEG0 LEN10)	; Read out the first result from QS970
12 QL1 = TONUMB (SRC_QSO)	; Convert alphanumeric value from QSO to a numerical value and assign it to QL0
13 QSO = SUBSTR (SRC_QS970 BEG11 LEN10)	; Read out the second result from QS970
14 QL2 = TONUMB (SRC_QSO)	; Convert alphanumeric value from QSO to a numerical value and assign it to QL2

Further information: User's Manual for **Klartext Programming** or **ISO Programming**

Log function

Once probing has finished, the control generates a log file in HTML format. The log file contains the results of the 3D deviation in graphical and tabular form. The control saves the log file in the same folder in which the NC program is located.

The log file contains the following data in the main axis, secondary axis and tool axis depending on the selected cycle (e.g., circle center point and diameter):

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- Upper and lower dimensions, as well as the determined deviation along the normal vector
- Measured actual coordinate
- Color coding of the values:
 - Green: Good
 - Orange: Rework
 - Red: Scrap

- Extrusion points:

The horizontal axis represents the direction for the extrusion. The blue points are the individual measuring points. The red lines indicate the lower limit and the upper limit of the dimensions. If a value violates a specified tolerance, the control will show the area in red color in the graphic.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If **Q1145 > 0** and **Q1146 = 0**, then the control will perform the number of extrusion points at the same position.
- When executing an extrusion with Cycles **1401 CIRCLE PROBING**, **1411 PROBING TWO CIRCLES** or **1404 PROBE SLOT/RIDGE**, the extrusion direction must equal **Q1140=+3**, otherwise the control will produce an error message.
- When defining the **TRANSER POSITION Q1120>0** within a touch probe cycle, the control will compensate the preset by the mean of deviations. The control calculates this mean from all measured extrusion points of the probing object according to the programmed **TRANSER POSITION Q1120**.

Example:

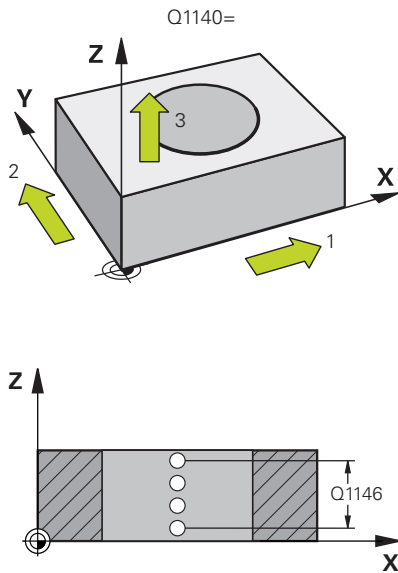
- Nominal position of touch point 1: 2.35 mm
- Results: **QS970** = 2.30000000 2.35000000 2.40000000
2.50000000

Mean: 2.387500000 mm

The preset is corrected by the mean from the nominal position, in this case by 0.0375 mm.

Cycle parameters

Help graphic



Parameter

Q1140 Direction for extrusion (1-3)?

- 1: Extrusion in the direction of the main axis
- 2: Extrusion in the direction of the secondary axis
- 3: Extrusion in the direction of the tool axis

Input: 1, 2, 3

Q1145 Number of extrusion points?

Number of measuring points that the cycle repeats over the length of the extrusion **Q1146**.

Input: 1...99

Q1146 Length of extrusion?

Length over which the measuring points are repeated.

Input: -99...+99

Q1149 Extrusion: Modal duration?

Effect of the cycle:

- 0: The extrusion is effective for only the next cycle.
- 1: The extrusion is effective until the end of the NC program.

Input: -99...+99

Example

11 TCH PROBE 1493 EXTRUSION PROBING ~	
Q1140=+3	;EXTRUSION DIRECTION ~
Q1145=+1	;EXTRUSION POINTS ~
Q1146=+0	;EXTRUSION LENGTH ~
Q1149=+0	;EXTRUSION MODAL

7.7 Calibrating a touch trigger probe

In order to precisely specify the actual trigger point of a 3D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.



Always calibrate a touch probe in the following cases:

- Commissioning
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, due to heating up of the machine
- Change of active tool axis

The control assumes the calibration values for the active probe system directly after the calibration process. The updated tool data are immediately effective. It is not necessary to repeat the tool call.

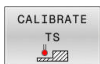
During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The control provides calibration cycles for calibrating the length and the radius:

Proceed as follows:



- ▶ Press the **TOUCH PROBE** key.



- ▶ Press the **CALIBRATE TS** soft key
- ▶ Select the calibration cycle

Calibration cycles provided by the control

Soft key	Function	Page
	Cycle 461 TS CALIBRATION OF TOOL LENGTH <ul style="list-style-type: none"> ■ Calibrating the length 	316
	Cycle 462 CALIBRATION OF A TS IN A RING <ul style="list-style-type: none"> ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge 	318
	Cycle 463 TS CALIBRATION ON STUD <ul style="list-style-type: none"> ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin 	321
	Cycle 460 CALIBRATION OF TS ON A SPHERE <ul style="list-style-type: none"> ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere 	324

7.8 Displaying calibration values

The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (main axis) and **CAL_OF2** (secondary axis). You can display the values on the screen by pressing the TOUCH PROBE TABLE soft key.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs. When running a touch probe cycle in the Manual Operation mode, the control saves the measuring log under the name TCHPRMAN.html. This file is stored in the folder TNC:*.



Ensure that the tool number in the tool table and the touch-probe number of the touch-probe table match. This is regardless of whether you want to use a touch-probe cycle in automatic mode or in **Manual operation** mode.



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

7.9 Cycle 461 TS CALIBRATION OF TOOL LENGTH

ISO programming

G461

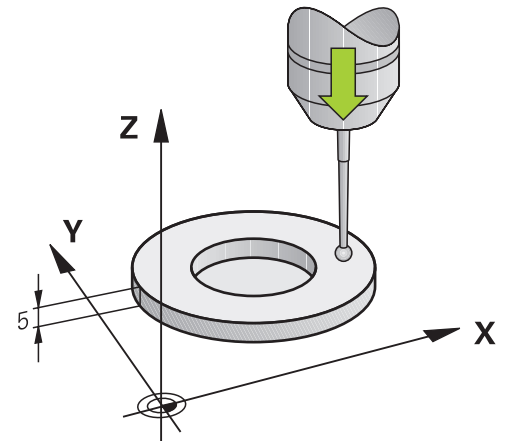
Application



Refer to your machine manual.

Before starting the calibration cycle, you must set the preset in the spindle axis so that $Z=0$ on the machine table; you must also pre-position the touch probe above the calibration ring.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.



Cycle sequence

- 1 The control orients the touch probe to the angle **CAL_ANG** specified in the touch probe table (only if your touch probe can be oriented).
- 2 The control probes from the current position in the negative spindle axis direction at the probing feed rate (column **F** from the touch probe table).
- 3 The control then retracts the touch probe at rapid traverse (column **FMAX** from the touch probe table) to the starting position.

Notes

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE**Danger of collision!**

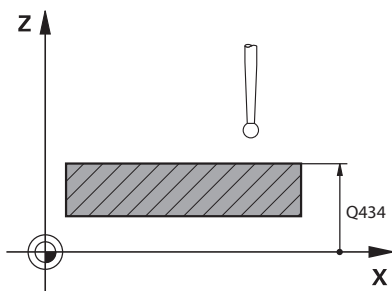
When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

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

Cycle parameters**Help graphic****Parameter****Q434 Preset for length?**

Preset for the length (e.g., height of the calibration ring). This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH ~

Q434=+5 ;PRESET

7.10 Cycle 462 CALIBRATION OF A TS IN A RING

ISO programming

G462

Application



Refer to your machine manual.

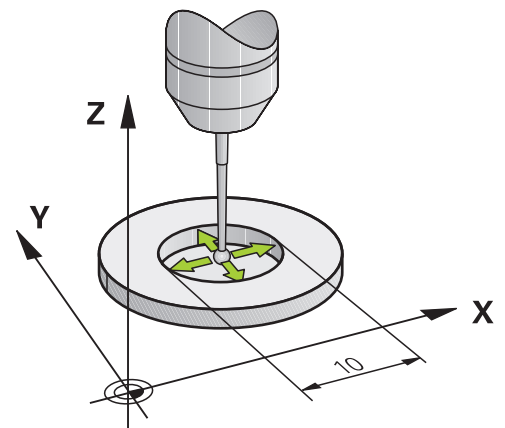
Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g., HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g., HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions".



Notes



In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

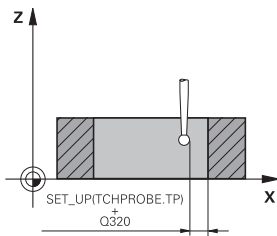
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q407 Radius of ring gauge?

Enter the radius of the ring gauge.

Input: **0.0001...99.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q423 Number of probes?

Number of measuring points on the diameter. This value has an absolute effect.

Input: **3...8**

Q380 Ref. angle in ref. axis?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **0...360**

Example

11 TCH PROBE 462 CALIBRATION OF A TS IN A RING ~

Q407=+5 ;RING RADIUS ~

Q320=+0 ;SET-UP CLEARANCE ~

Q423=+8 ;NO. OF PROBE POINTS ~

Q380=+0 ;REFERENCE ANGLE

7.11 Cycle 463 TS CALIBRATION ON STUD

ISO programming

G463

Application



Refer to your machine manual.

Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration pin. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration pin.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g., HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g., HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Notes

In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE**Danger of collision!**

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

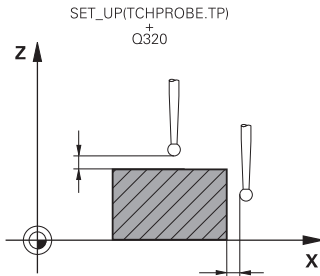
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q407 Radius of calibr. stud?

Diameter of the calibration stud

Input: **0.0001...99.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q423 Number of probes?

Number of measuring points on the diameter. This value has an absolute effect.

Input: **3...8**

Q380 Ref. angle in ref. axis?

Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

Input: **0...360**

Example

11 TCH PROBE 463 TS CALIBRATION ON STUD ~	
Q407=+5	;STUD RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q301=+1	;MOVE TO CLEARANCE ~
Q423=+8	;NO. OF PROBE POINTS ~
Q380=+0	;REFERENCE ANGLE

7.12 Cycle 460 CALIBRATION OF TS ON A SPHERE

ISO programming

G460

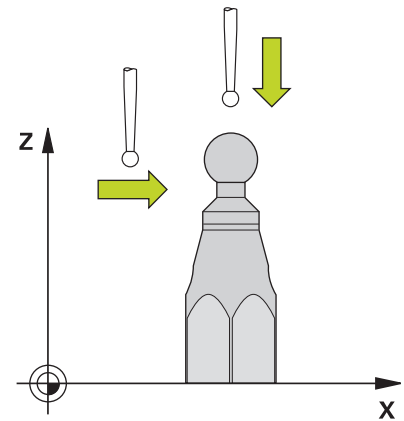
Application



Refer to your machine manual.

With Cycle **460** you can calibrate a triggering 3-D touch probe automatically using an exact calibration sphere.

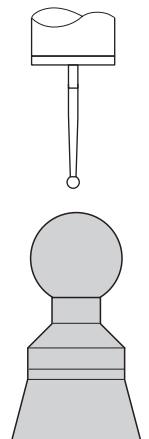
It is also possible to capture 3D calibration data. Option 92, **3D-ToolComp**, is required for this purpose. 3D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3D calibration data are stored under TNC:\system\3D-ToolComp*. The **DR2TABLE** column of the tool table references the 3DTC table. The 3D calibration data are then taken into account when probing. This 3D calibration is necessary if you want to achieve a very high accuracy with 3D Probing (e.g., Cycle **444**).



Before calibrating with a normal stylus:

Before starting the calibration cycle, you need to pre-position the touch probe:

- ▶ Define the approximate value of the radius R and length L of the touch probe
- ▶ In the working plane, center the touch probe above the calibration sphere
- ▶ Position the touch probe in the touch probe axis by approximately the amount of the set-up clearance above the calibration sphere. The set-up clearance consists of the value from the touch probe table plus the value from the cycle.



Pre-positioning with a normal stylus

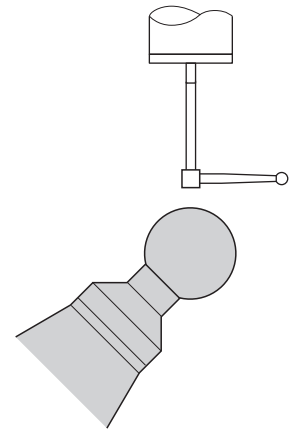
Before calibrating with an L-shaped stylus:

- ▶ Clamp the calibration sphere

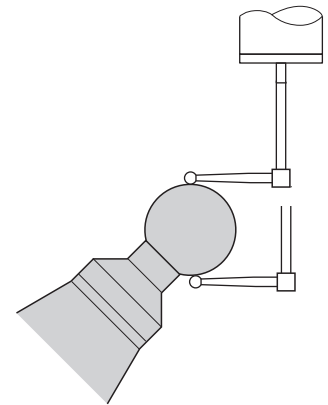


It must be possible to probe the north pole and south pole during calibration. If this is not possible, the control cannot determine the sphere radius. Ensure that no collision can occur.

- ▶ Define the approximate value of the radius **R** and length **L** of the touch probe. You can determine these with a tool presetter.
- ▶ Enter the approximate center offset in the touch probe table:
 - **CAL_OF1**: length of the extension
 - **CAL_OF2**: 0
- ▶ Insert the touch probe and orient it parallel to the main axis, for example with Cycle **13 ORIENTATION**
- ▶ Enter the calibration angle in the **CAL_ANG** column of the tool table.
- ▶ Position the center of the touch probe over the center of the calibration sphere
- ▶ Since the stylus is angled, the touch probe sphere is not centered over the calibration sphere.
- ▶ Position the touch probe in the tool axis by approximately the amount of the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere



Pre-positioning with an L-shaped stylus



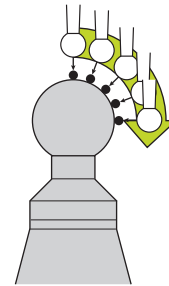
Calibration process with an L-shaped stylus

Cycle sequence

The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

Radius calibration Q433=0

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center
- 3 The first movement is in the plane, depending on the reference angle (**Q380**).
- 4 The controls positions the touch probe in the touch probe axis
- 5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
- 6 Once the equator has been determined, the determination of the spindle angle for calibration **CAL_ANG** begins (for L-shaped stylus)
- 7 Once **CAL_ANG** has been determined, the radius calibration begins
- 8 Finally, the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

**Radius and length calibration Q433=1**

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center
- 3 The first movement is in the plane, depending on the reference angle (**Q380**).
- 4 The control then positions the touch probe in touch-probe axis.
- 5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
- 6 Once the equator has been determined, the determination of the spindle angle for calibration **CAL_ANG** begins (for L-shaped stylus)
- 7 Once **CAL_ANG** has been determined, the radius calibration begins
- 8 The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.
- 9 The control determines the length of the touch probe at the north pole of the calibration sphere.
- 10 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

The setting in parameter **Q455** specifies whether you can perform an additional 3D calibration.

3D calibration Q455= 1...30

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 After calibration of the radius and length, the control retracts the touch probe in touch-probe axis. Then the control positions the touch probe above the north pole.
- 3 The probing process goes from the north pole to the equator in several steps. Deviations from the nominal value, and therefore the specific deflection behavior, are 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 3D calibration will be performed.
- 5 The deviations determined during the calibration are stored in a 3DTC table.
- 6 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.



- For an L-shaped stylus, the calibration takes place between the north and south pole.
- In order to calibrate the length, the position of the center point (**Q434**) of the calibration sphere relative to the active datum must be known. If this is not the case, then HEIDENHAIN recommends against using Cycle **460** to calibrate the length!
- One application example for calibrating the length with Cycle **460** is the comparison of two touch probes

Notes



HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

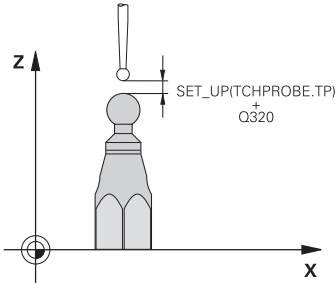
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- Depending on the accuracy of the pre-positioning, finding the equator of the calibration sphere will require a different number of touch points.
- In order to achieve optimum accuracy results with an L-shaped stylus, HEIDENHAIN recommends calibrating and probing at identical speeds. Note the setting of the feed override if it is active for probing.
- If you program **Q455=0**, the control will not perform a 3D calibration.
- If you program **Q455=1 to 30**, the control will perform a 3D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles. If you use Cycle **444**, you should first perform a 3D calibration.
- If you program **Q455=1 to 30**, a table will be stored under TNC: `\system\3D-ToolComp*`.
- If there is already a reference to a calibration table (entry in **DR2TABLE**), this table will be overwritten.
- If there is no reference to a calibration table (entry in **DR2TABLE**), then, in dependence of the tool number, a reference and the associated table will be created.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q407 Radius of calib. sphere?

Enter the exact radius of the calibration sphere being used.

Input: **0.0001...99.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q423 Number of probes?

Number of measuring points on the diameter. This value has an absolute effect.

Input: **3...8**

Q380 Ref. angle in ref. axis?

Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect.

Input: **0...360**

Q433 Calibrate length (0/1)?

Define whether the control will calibrate the touch probe length after radius calibration:

0: Do not calibrate touch probe length

1: Calibrate touch probe length

Input: **0, 1**

Q434 Preset for length?

Coordinate of the calibration sphere center. This value must be defined only if length calibration will be carried out. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q455 No. of points for 3-D calibrtn.?

Enter the number of touch points for 3D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3D calibration. During 3D calibration, the deflecting behavior of the touch probe is determined at various angles, and the values are stored in a table. 3D-ToolComp is required for 3D calibration.

Input: **0...30**

Example

11 TCH PROBE 460 TS CALIBRATION OF TS ON A SPHERE ~	
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q301=+1	;MOVE TO CLEARANCE ~
Q423=+4	;NO. OF PROBE POINTS ~
Q380=+0	;REFERENCE ANGLE ~
Q433=+0	;CALIBRATE LENGTH ~
Q434=-2.5	;PRESET ~
Q455=+15	;NO. POINTS 3-D CAL.

8

**Touch Probe
Cycles: Automatic
Kinematics
Measurement**

8.1 Kinematics measurement with TS touch probes (option 48)

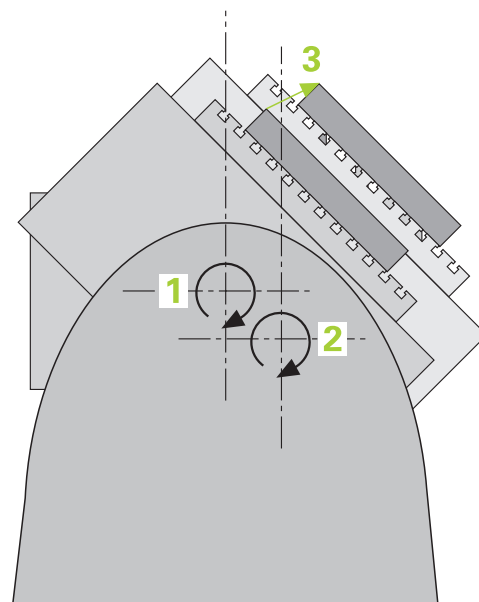
Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure) and the kinematic conditions actually existing on the machine (see **2** in the figure). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure). It is therefore necessary for the model to approach reality as closely as possible.





The **KinematicsOpt** function of the control is an important component that helps you meet these complex requirements in real life: a 3D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, a calibration sphere is attached at any position on the machine table, and measured with a resolution that you define. During cycle definition, you simply define for each rotary axis the area that you want to measure.

From the measured values, the control calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics table.



Overview

The control offers the following cycles that enable you to automatically save, restore, check, and optimize the machine kinematics:

Soft key	Cycle	Page
	Cycle 450 SAVE KINEMATICS (option 48) <ul style="list-style-type: none"> ■ Storing the active machine kinematic configuration ■ Restoring previously saved kinematic configuration 	336
	Cycle 451 MEASURE KINEMATICS (option 48) <ul style="list-style-type: none"> ■ Automatic checking of the machine kinematic configuration ■ Optimizing the machine kinematic configuration 	339
	Cycle 452 PRESET COMPENSATION (option 48) <ul style="list-style-type: none"> ■ Automatic checking of the machine kinematic configuration ■ Optimizing the kinematic transformation chain of the machine 	357
	Cycle 453 KINEMATICS GRID (option 48) <ul style="list-style-type: none"> ■ Automatic checking depending on the tilting axis position of the machine kinematic configuration ■ Optimizing the machine kinematic configuration 	370

8.2 Requirements



Refer to your machine manual.

Advanced Function Set 1 (option 8) must have been enabled.

Option 48 must have been enabled.

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

The following are requirements for using the KinematicsOpt option:

- The 3D touch probe used for the measurement must be calibrated.
- The cycles can only be carried out with the tool axis Z.
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table
- The kinematics description of the machine must be complete and correct, and the transformation dimensions must have been entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine manufacturer during commissioning).
- The machine manufacturer must have defined the machine parameters for **CfgKinematicsOpt** (no. 204800) in the configuration data.
 - **maxModification** (no. 204801) specifies the tolerance limit starting from which the control is to display a message if the changes made to the kinematic data exceed this limit value.
 - **maxDevCalBall** (no. 204802) defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter
 - **mStrobeRotAxPos** (no. 204803) defines an M function that is specifically configured by the machine manufacturer and is used to position the rotary axes.



HEIDENHAIN recommends using the calibration spheres **KKH 250 (ID number 655475-01)** or **KKH 80 (ID number 655475-03)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

Notes



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

NOTICE

Danger of collision!

When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

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

NOTICE

Danger of collision!

A change in the kinematics always changes the preset as well. Basic rotations will automatically be reset to 0. There is a danger of collision!

- ▶ After an optimization, reset the preset

Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the position of the rotary axes. If an M function has been defined in the machine parameter, you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for **450**).
- If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise, the changes could be lost in certain circumstances.

8.3 Cycle 450 SAVE KINEMATICS (option 48)

ISO programming

G450

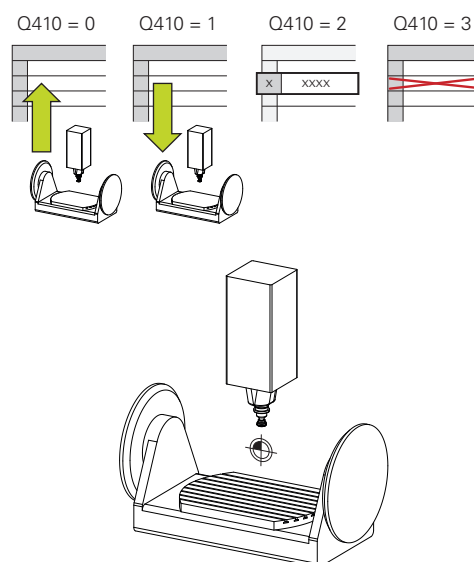
Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

With touch probe cycle **450** you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.



Notes



Only save and restore data with Cycle **450** while no tool carrier kinematics configuration that includes transformations is active.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Always save the active kinematic model before running a kinematics optimization.
Advantage:
 - You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g., power failure).
- With the **Restore** mode, note the following:
 - The control can restore saved data only to a matching kinematic configuration
 - A change in the kinematics always changes the preset as well. So redefine the preset, if required.
- The cycle does not restore identical values. It only restores values that differ from the present values. Compensations can only be restored if they had been saved before.

Cycle parameters

Help graphic	Parameter
	<p>Q410 Mode (0/1/2/3)? Define whether a kinematic model will be saved or restored:</p> <p>0: Save active kinematics 1: Restore saved kinematics 2: Display the current memory status 3: Delete a data record</p> <p>Input: 0, 1, 2, 3</p>
	<p>Q409/QS409 Name of data record? Number or name of data record identifier. Q409 does not function if mode 2 has been selected. Wildcards can be used for searches in modes 1 and 3 (Restore and Delete). If the control finds several possible data records because of the wildcards, the control restores the mean values of the data (mode 1) or deletes all selected data records after confirmation (mode 3). You can use the following wildcards in searches:</p> <p>?: A single, undefined character \$: A single alphabetic character (letter) #: A single, undefined number *: An undefined string of any length</p> <p>Input: 0...99999 or max. 255 characters. A total of 16 memory locations are available.</p>

Saving the current kinematics

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+0 ;MODE ~
Q409=+947 ;MEMORY DESIGNATION

Restoring data records

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+1 ;MODE ~
Q409=+948 ;MEMORY DESIGNATION

Displaying all saved data records

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+2 ;MODE ~
Q409=+949 ;MEMORY DESIGNATION

Deleting data records

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+3 ;MODE ~
Q409=+950 ;MEMORY DESIGNATION

Log function

After running Cycle **450**, the control creates a log (**TCHPRAUTO.html**) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

The other data in the log vary depending on the selected mode:

- Mode 0: Logging of all axis entries and transformation entries of the kinematics chain that the control has saved.
- Mode 1: Logging of all transformation entries before and after restoring the kinematics configuration.
- Mode 2: List of the saved data records
- Mode 3: List of the deleted data records

Notes on data management

The control stores the saved data in the file **TNC:\table\DATA450.KD**. This file can be backed up to an external PC with **TNCremo**, for example. If you delete the file, the stored data are removed, too. If the data in the file are changed manually, the data records may become corrupted so that they are unusable.



Operating notes:

- If the file **TNC:\table\DATA450.KD** does not exist, it is generated automatically when Cycle **450** is run.
- Make sure that you delete any empty files with the name **TNC:\table\DATA450.KD** before starting Cycle **450**. If there is an empty memory table (**TNC:\table\DATA450.KD**) without any rows in it, an error message will be issued when running Cycle **450**. In this case, delete the empty memory table and call the cycle again.
- Do not change stored data manually.
- Make a backup of the **TNC:\table\DATA450.KD** file so that you can restore the file, if necessary (e.g. if the data medium is damaged).

8.4 Cycle 451 MEASURE KINEMATICS (option 48)

ISO programming
G451

Application

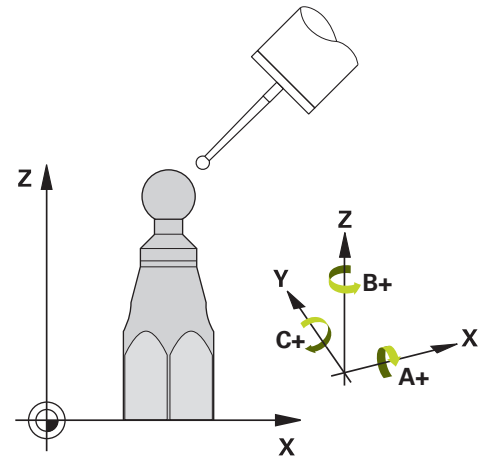


Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

Touch probe cycle **451** enables you to check and, if required, optimize the kinematics of your machine. Use the 3D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

The control will determine the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.



Cycle run

- 1 Clamp the calibration sphere and check for potential collisions.
- 2 In the Manual Operation operating mode, set the preset to the center of the sphere or, if you defined **Q431** = 1 or **Q431** = 3: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
- 3 Select the Program Run operating mode and start the calibration program.
- 4 The control automatically measures all rotary axes successively in the resolution you defined.



Programming and operating notes:

- If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC Start**.
- During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

Result parameter Q

The control saves the results of the touch probe cycle in the following Q parameters:

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (-1 if axis was not measured)
Q142	Standard deviation measured in the B axis (-1 if axis was not measured)
Q143	Standard deviation measured in the C axis (-1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (-1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (-1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (-1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Result parameter QS

The control saves the measured position faults of rotary axes in the QS parameters **QS144 - QS146**. Each result is ten characters long. The results are separated from each other by a space.

Example: **QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"**

Q parameter number	Meaning
QS144	Position error of A axis $E_{Y0A} E_{Z0A} E_{B0A} E_{C0A}$
QS145	Position error of B axis $E_{Z0B} E_{X0B} E_{C0B} E_{A0B}$
QS146	Position error of C axis $E_{X0C} E_{Y0C} E_{A0C} E_{B0C}$



Position faults are deviations from the ideal axis position and are marked by four characters.

Example: E_{X0C} = Position error of the C axis in X direction.

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

The cycle produces the following results within the QS parameter **QS146**:

QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"

The example below shows how to convert the results produced into numerical values.

11 QS0 = SUBSTR (SRC_QS146 BEG0 LEN10)	; Read out the first result E_{X0C} from QS146
12 QL0 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL0
13 QS0 = SUBSTR (SRC_QS146 BEG11 LEN10)	; Read out the second result E_{Y0C} from QS146
14 QL1 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL1
15 QS0 = SUBSTR (SRC_QS146 BEG22 LEN10)	; Read out the third result E_{A0C} from QS146
16 QL2 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL2
17 QS0 = SUBSTR (SRC_QS146 BEG33 LEN10)	; Read out the forth result E_{B0C} from QS146
18 QL3 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL3

Further information: User's Manual for **Klartext Programming** or **ISO Programming**

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 = $(-90^\circ - +90^\circ) / (4 - 1) = -60^\circ$
 - 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 = $(270^\circ - 90^\circ) / (4 - 1) = +60^\circ$
 - Measuring point 1 = +90°
 - Measuring point 2 = +150°
 - Measuring point 3 = +210°
 - Measuring point 4 = +270°

Machines with Hirth-coupled axes

NOTICE

Danger of collision!

In order to be positioned, the axis must move out of the Hirth grid. If necessary, the control rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points). There is a danger of collision!

- ▶ So remember to leave a large enough set-up clearance to prevent any risk of collision between the touch probe and calibration sphere
- ▶ Also ensure that there is enough space to reach the set-up clearance (software limit switch)

NOTICE

Danger of collision!

Depending on the machine configuration, the control cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine manufacturer, enabling the control to move the rotary axes. The machine manufacturer must have entered the number of the M function in the machine parameter **mStrokeRotAxPos** (no. 204803) for this purpose. There is a danger of collision!

- ▶ Note the documentation of the machine manufacturer



- Define the retracting height above 0 if software option 9 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 = $(\mathbf{Q412} - \mathbf{Q411}) / (\mathbf{Q414} - 1)$

Calculated stepping angle = $(90^\circ - (-30^\circ)) / (4 - 1) = 120 / 3 = 40^\circ$

Measuring position 1 = **Q411** + 0 * stepping angle = -30° → -30°

Measuring position 2 = **Q411** + 1 * stepping angle = +10° → 9°

Measuring position 3 = **Q411** + 2 * stepping angle = +50° → 51°

Measuring position 4 = **Q411** + 3 * stepping angle = +90° → 90°

Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1 or 2), for example when commissioning the machine.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°. Thus, define a starting angle of 90° and an end angle of 270°.

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the **Check** mode.

i If a measuring point has been defined at 0°, it will be ignored because the reference measurement is always done at 0°.

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machines with rotary tables/tilting tables: Clamp the calibration sphere as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

i Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Notes on the accuracy



If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine manual provides further information.

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion output by the control in the measurement log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.



If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
 - Number of measuring points between 1 and 2
 - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
 - Number of measuring points between 3 and 6
 - The start and end angles should cover the largest possible traverse range of the rotary axes.
 - Position the calibration sphere in such a way on the machine table that, with rotary table axes, there is a large measuring circle or that, on swivel head axes, measurement can be made at a representative position (e.g., in the center of the traverse range).
- **Optimization of a specific rotary axis position**
 - Number of measuring points between 2 and 3
 - The measurements are made with the aid of the inclination angle of an axis (**Q413/Q417/Q421**) around the rotary axis angle at which the workpiece is to be machined later.
 - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.
- **Inspecting the machine accuracy**
 - Number of measuring points between 4 and 8
 - The start and end angles should cover the largest possible traverse range of the rotary axes.
- **Determination of the rotary axis backlash**
 - Number of measuring points between 8 and 12
 - The start and end angles should cover the largest possible traverse range of the rotary axes.

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is performed with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432**, you can activate backlash measurement. Enter an angle that the control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.



Backlash measurement is not possible if an M function for positioning the rotary axes is set in the optional **mStrobeRotAxPos** machine parameter (no. 204803) or if the axis is a Hirth axis.



Programming and operating notes:

- The control does not perform an automatic backlash compensation.
- If the measuring circle radius is < 1 mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can ascertain the rotary axis backlash.

Further information: "Log function", Page 356

Notes



Angle compensation is possible only with option 52 KinematicsComp.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
 - ▶ Set the preset and the basic rotation again after optimization.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
 - Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
 - As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
 - Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or set input parameter **Q431** to 1 or 3, respectively.
 - For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
 - The control ignores cycle definition data that applies to inactive axes.
 - A correction in the machine datum (**Q406=3**) is only possible if superimposed rotary axes on the spindle head side or table side are measured.
 - If you have activated presetting before the calibration (**Q431 = 1/3**), then move the touch probe to the set-up clearance (**Q320 + SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.
 - Programming in inches: The control always records the log data and results of measurement in millimeters.
 - After measuring the kinematics, you must re-determine the preset.

Notes about machine parameters

- If the optional machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to -1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0° .
- In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.
- For angle optimization, the machine manufacturer must adapt the configuration correspondingly.

Cycle parameters

Help graphic	Parameter
	<p>Q406 Mode (0/1/2/3)?</p> <p>Define whether the control will check or optimize the active kinematics:</p> <p>0: Check the active machine kinematics. The control measures the kinematics in the rotary axes you have defined, but it does not make any changes to the active kinematics. The control displays the measurement results in a measuring log.</p> <p>1: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes the rotary axes positions of the active kinematics.</p> <p>2: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes angle and position errors. KinematicsComp Option 52 is the prerequisite for angle error compensation.</p> <p>3: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then automatically compensates the machine datum. It then optimizes angle and position errors. KinematicsComp Option 52 is the prerequisite.</p> <p>Input: 0, 1, 2, 3</p>
	<p>Q407 Radius of calib. sphere?</p> <p>Enter the exact radius of the calibration sphere being used.</p> <p>Input: 0.0001...99.9999</p>
	<p>Q320 Set-up clearance?</p> <p>Additional distance between touch point and ball tip. Q320 is active in addition to the SET_UP column in the touch probe table. This value has an incremental effect.</p> <p>Input: 0...99999.9999 or PREDEF</p>
	<p>Q408 Retraction height?</p> <p>0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C.</p> <p>> 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253. This value has an absolute effect.</p> <p>Input: 0...99999.9999</p>

Help graphic

Parameter

Q253 Feed rate for pre-positioning?

Define the traversing speed of the tool during pre-positioning in mm/min.

Input: **0...99999.9999** or **FMAX, FAUTO, PREDEF**

Q380 Ref. angle in ref. axis?

Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect.

Input: **0...360**

Q411 Starting angle in A axis?

Starting angle in the A axis at which the first measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q412 End angle in A axis?

End angle in the A axis at which the last measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q413 Angle of incidence in A axis?

Angle of incidence in the A axis at which the other rotary axes will be measured.

Input: **-359.9999...+359.9999**

Q414 No. of meas. points in A (0...12)?

Number of measuring points the control will use to measure the A axis.

If the input value = 0, the control does not measure the respective axis.

Input: **0...12**

Q415 Starting angle in B axis?

Starting angle in the B axis at which the first measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q416 End angle in B axis?

End angle in the B axis at which the last measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q417 Angle of incidence in B axis?

Angle of incidence in the B axis at which the other rotary axes will be measured.

Input: **-359.999...+360.000**

Help graphic

Parameter

Q418 No. of meas. points in B (0...12)?

Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis.

Input: **0...12**

Q419 Starting angle in C axis?

Starting angle in the C axis at which the first measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q420 End angle in C axis?

End angle in the C axis at which the last measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q421 Angle of incidence in C axis?

Angle of incidence in the C axis at which the other rotary axes will be measured.

Input: **-359.9999...+359.9999**

Q422 No. of meas. points in C (0...12)?

Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis.

Input: **0...12**

Q423 Number of probes?

Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision.

Input: **3...8**

Q431 Preset (0/1/2/3)?

Define whether the control will automatically set the active preset at the center of the sphere:

0: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle

1: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

2: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle

3: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

Input: **0, 1, 2, 3**

Help graphic

Parameter

Q432 Angular range of backlash comp.?

Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash.

Input: **-3...+3**

Saving and checking the kinematics

11	TOOL CALL "TOUCH_PROBE" Z
12	TCH PROBE 450 SAVE KINEMATICS ~
	Q410=+0 ;MODE ~
	Q409=+5 ;MEMORY DESIGNATION
13	TCH PROBE 451 MEASURE KINEMATICS ~
	Q406=+0 ;MODE ~
	Q407=+12.5 ;SPHERE RADIUS ~
	Q320=+0 ;SET-UP CLEARANCE ~
	Q408=+0 ;RETR. HEIGHT ~
	Q253=+750 ;F PRE-POSITIONING ~
	Q380=+0 ;REFERENCE ANGLE ~
	Q411=-90 ;START ANGLE A AXIS ~
	Q412=+90 ;ENDWINKEL A-ACHSE ~
	Q413=+0 ;INCID. ANGLE A AXIS ~
	Q414=+0 ;MEAS. POINTS A AXIS ~
	Q415=-90 ;START ANGLE B AXIS ~
	Q416=+90 ;END ANGLE B AXIS ~
	Q417=+0 ;INCID. ANGLE B AXIS ~
	Q418=+2 ;MEAS. POINTS B AXIS ~
	Q419=-90 ;START ANGLE C AXIS ~
	Q420=+90 ;END ANGLE C AXIS ~
	Q421=+0 ;INCID. ANGLE C AXIS ~
	Q422=+2 ;MEAS. POINTS C AXIS ~
	Q423=+4 ;NO. OF PROBE POINTS ~
	Q431=+0 ;PRESET ~
	Q432=+0 ;BACKLASH, ANG. RANGE

Various modes (Q406)

Test mode Q406 = 0

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control records the results of a possible position optimization but does not make any adjustments.

"Optimize position of rotary axes" mode Q406 = 1

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the control tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data are adjusted automatically.

"Optimize position and angle" 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)
- After that, the position is optimized. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.



Depending on the machine kinematics for correctly determining the angles, HEIDENHAIN recommends performing the measurement once with an inclination angle of 0°.

"Optimize machine datum, position, and angle" mode (Q406 = 3)

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control automatically tries to optimize the machine datum (option 52). In order to use a machine datum to compensate for the angular position of a rotary axis, the rotary axis to be corrected must be nearer to the machine base in the machine kinematics than the measured rotary axis.
- Then the control tries to optimize the angular orientation of the rotary axis by means of compensation (option 52).
- After that, the position is optimized. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.



- For correct determination of the angular position errors, HEIDENHAIN recommends setting the affected rotary axis to an inclination angle of 0° for this measurement.
- After correcting a machine datum, the control tries to reduce the compensation of the associated angular position error (**locErrA/locErrB/locErrC**) of the measured rotary axis.

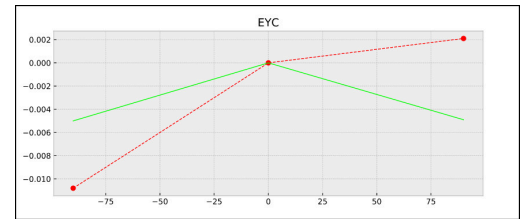
Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

11	TOOL CALL "TOUCH_PROBE" Z
12	TCH PROBE 451 MEASURE KINEMATICS ~
Q406	=+1 ;MODE ~
Q407	=+12.5 ;SPHERE RADIUS ~
Q320	=+0 ;SET-UP CLEARANCE ~
Q408	=+0 ;RETR. HEIGHT ~
Q253	=+750 ;F PRE-POSITIONING ~
Q380	=+0 ;REFERENCE ANGLE ~
Q411	=-90 ;START ANGLE A AXIS ~
Q412	=+90 ;END ANGLE A AXIS ~
Q413	=+0 ;INCID. ANGLE A AXIS ~
Q414	=+0 ;MEAS. POINTS A AXIS ~
Q415	=-90 ;START ANGLE B AXIS ~
Q416	=+90 ;END ANGLE B AXIS ~
Q417	=+0 ;INCID. ANGLE B AXIS ~
Q418	=+4 ;MEAS. POINTS B AXIS ~
Q419	=+90 ;START ANGLE C AXIS ~
Q420	=+270 ;END ANGLE C AXIS ~
Q421	=+0 ;INCID. ANGLE C AXIS ~
Q422	=+3 ;MEAS. POINTS C AXIS ~
Q423	=+3 ;NO. OF PROBE POINTS ~
Q431	=+1 ;PRESET ~
Q432	=+0.5 ;BACKLASH, ANG. RANGE

Log function

After running Cycle 451, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Tool name
- Active kinematics
- Mode used (0=Check/1=Optimize position/2=Optimize pose/3=Optimize machine datum and pose)
- Inclination angles
- For each measured rotary axis:
 - Starting angle
 - End angle
 - Number of measuring points
 - Measuring circle radius
 - Averaged backlash, if **Q423>0**
 - Positions of the axes
 - Angular orientation errors only with **KinematicsComp** (software option 52)
 - Standard deviation (scatter)
 - Maximum deviation
 - Angular error
 - Compensation values in all axes (preset shift)
 - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Averaged positioning error and standard deviation of the positioning errors to 0
 - SVG files with graphs: measured and optimized errors of individual measurement positions.
 - Red curve: measured positions
 - Green curve: optimized values after cycle has run
 - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = component error in Y of axis C)
 - X axis of the graph: rotary axis position in degrees
 - Y axis of the graph: position deviations in mm



Sample measurement: EYC component error in Y of axis C

8.5 Cycle 452 PRESET COMPENSATION (option 48)

ISO programming
G452

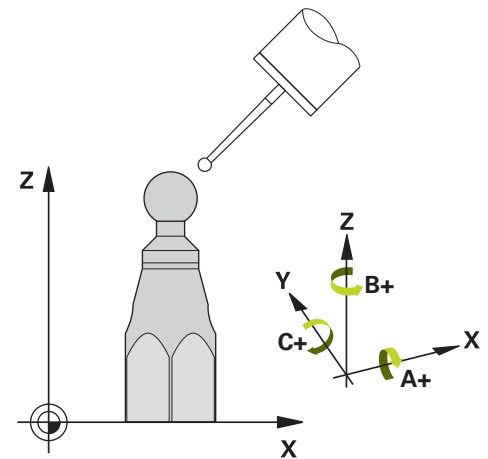
Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

Touch probe cycle **452** optimizes the kinematic transformation chain of your machine (see "Cycle 451 MEASURE KINEMATICS (option 48)", Page 339). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is at the center of the calibration sphere after optimization.



Cycle run



Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

- 1 Clamp the calibration sphere
- 2 Measure the complete reference head with Cycle **451**, and then use Cycle **451** to set the preset in the center of the sphere.
- 3 Insert the second head
- 4 Use Cycle **452** to measure the interchangeable head up to the point where the head is changed.
- 5 Use Cycle **452** to adjust other interchangeable heads to the reference head

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

- 1 Clamp the calibration sphere and check for potential collisions.
- 2 Set the preset in the calibration sphere.
- 3 Set the preset on the workpiece, and start machining the workpiece.
- 4 Use Cycle **452** for preset compensation at regular intervals. The control measures the drift of the axes involved and compensates it in the kinematics description.

Result parameter Q

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (-1 if axis was not measured)
Q142	Standard deviation measured in the B axis (-1 if axis was not measured)
Q143	Standard deviation measured in the C axis (-1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (-1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (-1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (-1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Result parameter QS

The control saves the measured position faults of rotary axes in the QS parameters **QS144 - QS146**. Each result is ten characters long. The results are separated from each other by a space.

Example: **QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"**

Q parameter number	Meaning
QS144	Position error of A axis $E_{Y0A} E_{Z0A} E_{B0A} E_{C0A}$
QS145	Position error of B axis $E_{Z0B} E_{X0B} E_{C0B} E_{A0B}$
QS146	Position error of C axis $E_{X0C} E_{Y0C} E_{A0C} E_{B0C}$



Position faults are deviations from the ideal axis position and are marked by four characters.

Example: E_{X0C} = Position error of the C axis in X direction.

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

The cycle produces the following results within the QS parameter **QS146**:

QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"

The example below shows how to convert the results produced into numerical values.

11 <code>QS0 = SUBSTR (SRC_QS146 BEG0 LEN10)</code>	; Read out the first result E_{X0C} from QS146
12 <code>QL0 = TONUMB (SRC_QS0)</code>	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL0
13 <code>QS0 = SUBSTR (SRC_QS146 BEG11 LEN10)</code>	; Read out the second result E_{Y0C} from QS146
14 <code>QL1 = TONUMB (SRC_QS0)</code>	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL1
15 <code>QS0 = SUBSTR (SRC_QS146 BEG22 LEN10)</code>	; Read out the third result E_{A0C} from QS146
16 <code>QL2 = TONUMB (SRC_QS0)</code>	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL2
17 <code>QS0 = SUBSTR (SRC_QS146 BEG33 LEN10)</code>	; Read out the forth result E_{B0C} from QS146
18 <code>QL3 = TONUMB (SRC_QS0)</code>	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL3

Further information: User's Manual for **Klartext Programming** or **ISO Programming**

Notes



In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
 - ▶ Set the preset and the basic rotation again after optimization.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
 - Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
 - As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
 - Ensure that all functions for tilting the working plane are reset.
 - Before defining the cycle, you must set the preset at the center of the calibration sphere and activate it.
 - For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse an angle of 1° to the limit switch. The control needs this traverse for internal backlash compensation.
 - For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while touch probe monitoring is inactive.
 - Programming in inches: The control always records the log data and results of measurement in millimeters.



- If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle **450**, so that in case of a failure the most recently active kinematic configuration can be restored.

Notes about machine parameters

- In the machine parameter **maxModification** (no. 204801), the machine manufacturer defines the permissible limit value for modifications of a transformation. If the kinematics data determined exceed the permissible limit value, the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC Start**.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	<p>Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: 0.0001...99.9999</p>
	<p>Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is active in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF</p>
	<p>Q408 Retraction height? 0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. > 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253. This value has an absolute effect. Input: 0...99999.9999</p>
	<p>Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: 0...99999.9999 or FMAX, FAUTO, PREDEF</p>
	<p>Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: 0...360</p>
	<p>Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999</p>
	<p>Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999</p>
	<p>Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999</p>

Help graphic**Parameter****Q414 No. of meas. points in A (0...12)?**

Number of measuring points the control will use to measure the A axis.

If the input value = 0, the control does not measure the respective axis.

Input: **0...12**

Q415 Starting angle in B axis?

Starting angle in the B axis at which the first measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q416 End angle in B axis?

End angle in the B axis at which the last measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q417 Angle of incidence in B axis?

Angle of incidence in the B axis at which the other rotary axes will be measured.

Input: **-359.999...+360.000**

Q418 No. of meas. points in B (0...12)?

Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis.

Input: **0...12**

Q419 Starting angle in C axis?

Starting angle in the C axis at which the first measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q420 End angle in C axis?

End angle in the C axis at which the last measurement will be made. This value has an absolute effect.

Input: **-359.9999...+359.9999**

Q421 Angle of incidence in C axis?

Angle of incidence in the C axis at which the other rotary axes will be measured.

Input: **-359.9999...+359.9999**

Q422 No. of meas. points in C (0...12)?

Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis.

Input: **0...12**

Q423 Number of probes?

Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision.

Input: **3...8**

Help graphic

Parameter

Q432 Angular range of backlash comp.?

Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash.

Input: **-3...+3**

Calibration program

11	TOOL CALL "TOUCH_PROBE" Z
12	TCH PROBE 450 SAVE KINEMATICS ~
Q410	=+0 ;MODE ~
Q409	=+5 ;MEMORY DESIGNATION
13	TCH PROBE 452 PRESET COMPENSATION ~
Q407	=+12.5 ;SPHERE RADIUS ~
Q320	=+0 ;SET-UP CLEARANCE ~
Q408	=+0 ;RETR. HEIGHT ~
Q253	=+750 ;F PRE-POSITIONING ~
Q380	=+0 ;REFERENCE ANGLE ~
Q411	=-90 ;START ANGLE A AXIS ~
Q412	=+90 ;END ANGLE A AXIS ~
Q413	=+0 ;INCID. ANGLE A AXIS ~
Q414	=+0 ;MEAS. POINTS A AXIS ~
Q415	=-90 ;START ANGLE B AXIS ~
Q416	=+90 ;END ANGLE B AXIS ~
Q417	=+0 ;INCID. ANGLE B AXIS ~
Q418	=+2 ;MEAS. POINTS B AXIS ~
Q419	=-90 ;START ANGLE C AXIS ~
Q420	=+90 ;END ANGLE C AXIS ~
Q421	=+0 ;INCID. ANGLE C AXIS ~
Q422	=+2 ;MEAS. POINTS C AXIS ~
Q423	=+4 ;NO. OF PROBE POINTS ~
Q432	=+0 ;BACKLASH, ANG. RANGE

Adjustment of interchangeable heads



The head change function can vary depending on the individual machine tool. Refer to your machine manual.

- ▶ Load the second interchangeable head.
- ▶ Insert the touch probe
- ▶ Measure the interchangeable head with Cycle **452**
- ▶ Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with **Q422**)
- ▶ The preset and the position of the calibration sphere must not be changed during the entire process.
- ▶ All other interchangeable heads can be adjusted in the same way

Adjusting an interchangeable head

11 TOOL CALL "TOUCH_PROBE" Z	
12 TCH PROBE 452 PRESET COMPENSATION ~	
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+2000	;F PRE-POSITIONING ~
Q380=+45	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+45	;INCID. ANGLE A AXIS ~
Q414=+4	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+2	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+0	;MEAS. POINTS C AXIS ~
Q423=+4	;NO. OF PROBE POINTS ~
Q432=+0	;BACKLASH, ANG. RANGE

The goal of this procedure is that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, the adjustment of a fork head with A and C axes is described. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- ▶ Insert the interchangeable head that will be used as a reference head.
- ▶ Clamp the calibration sphere
- ▶ Insert the touch probe
- ▶ Use Cycle **451** to measure the complete kinematics, including the reference head
- ▶ Define the preset (using **Q431** = 2 or 3 in Cycle **451**) after measuring the reference head

Measuring a reference head

11 TOOL CALL "TOUCH_PROBE" Z	
12 TCH PROBE 451 MEASURE KINEMATICS ~	
Q406=+1	;MODE ~
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+2000	;F PRE-POSITIONING ~
Q380=+45	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+45	;INCID. ANGLE A AXIS ~
Q414=+4	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+2	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+3	;MEAS. POINTS C AXIS ~
Q423=+4	;NO. OF PROBE POINTS ~
Q431=+3	;PRESET ~
Q432=+0	;BACKLASH, ANG. RANGE

Drift compensation

i This procedure can also be performed on machines without rotary axes.

During machining, various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle **452**.

- ▶ Clamp the calibration sphere
- ▶ Insert the touch probe
- ▶ Measure the complete kinematics with Cycle **451** before starting the machining process
- ▶ Define the preset (using **Q432** = 2 or 3 in Cycle **451**) after measuring the kinematics
- ▶ Then set the presets on your workpiece and start the machining process.

Reference measurement for drift compensation

11	TOOL CALL "TOUCH_PROBE" Z
12	CYCL DEF 247 PRESETTING ~
Q339	=+1 ;PRESET NUMBER
13	TCH PROBE 451 MEASURE KINEMATICS ~
Q406	=+1 ;MODE ~
Q407	=+12.5 ;SPHERE RADIUS ~
Q320	=+0 ;SET-UP CLEARANCE ~
Q408	=+0 ;RETR. HEIGHT ~
Q253	=+750 ;F PRE-POSITIONING ~
Q380	=+45 ;REFERENCE ANGLE ~
Q411	=+90 ;START ANGLE A AXIS ~
Q412	=+270 ;END ANGLE A AXIS ~
Q413	=+45 ;INCID. ANGLE A AXIS ~
Q414	=+4 ;MEAS. POINTS A AXIS ~
Q415	=-90 ;START ANGLE B AXIS ~
Q416	=+90 ;END ANGLE B AXIS ~
Q417	=+0 ;INCID. ANGLE B AXIS ~
Q418	=+2 ;MEAS. POINTS B AXIS ~
Q419	=+90 ;START ANGLE C AXIS ~
Q420	=+270 ;END ANGLE C AXIS ~
Q421	=+0 ;INCID. ANGLE C AXIS ~
Q422	=+3 ;MEAS. POINTS C AXIS ~
Q423	=+4 ;NO. OF PROBE POINTS ~
Q431	=+3 ;PRESET ~
Q432	=+0 ;BACKLASH, ANG. RANGE

- ▶ Measure the drift of the axes at regular intervals.
- ▶ Insert the touch probe
- ▶ Activate the preset in the calibration sphere.
- ▶ Use Cycle **452** to measure the kinematics.
- ▶ The preset and the position of the calibration sphere must not be changed during the entire process.

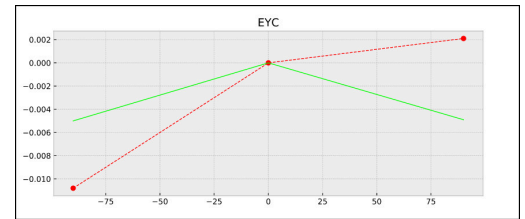
Drift compensation

11 TOOL CALL "TOUCH_PROBE" Z	
13 TCH PROBE 452 PRESET COMPENSATION ~	
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+9999	;F PRE-POSITIONING ~
Q380=+45	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+45	;INCID. ANGLE A AXIS ~
Q414=+4	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+2	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+3	;MEAS. POINTS C AXIS ~
Q423=+3	;NO. OF PROBE POINTS ~
Q432=+0	;BACKLASH, ANG. RANGE

Log function

After running Cycle **452**, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Tool name
- Active kinematics
- Mode used
- Inclination angles
- For each measured rotary axis:
 - Starting angle
 - End angle
 - Number of measuring points
 - Measuring circle radius
 - Averaged backlash, if **Q423>0**
 - Positions of the axes
 - Standard deviation (scatter)
 - Maximum deviation
 - Angular error
 - Compensation values in all axes (preset shift)
 - Position before preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Position after preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Averaged positioning error
 - SVG files with graphs: measured and optimized errors of individual measurement positions.
 - Red curve: measured positions
 - Green curve: optimized values
 - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = deviations) of the Y axis in dependency of the C axis.
 - X axis of the graph: rotary axis position in degrees
 - Y axis of the graph: position deviations in mm



Sample measurement: EYC deviations of the Y axis in dependency of the C axis

8.6 Cycle 453 KINEMATICS GRID (option 48)

ISO programming

G453

Application



Refer to your machine manual.

KinematicsOpt (software option 48) is required.

This function must be enabled and adapted by the machine manufacturer.

To use this cycle, your machine manufacturer needs to create and configure a compensation table (*.kco) first and enter some more settings.

Even if your machine was already optimized regarding positioning errors (e.g., via Cycle 451), residual errors at the Tool Center Point (TCP) during tilting of the rotary axes may remain. These can result, for example, from component errors (e.g., a bearing error) with head rotation axes.

Cycle 453 KINEMATICS GRID enables errors in swivel heads to be detected and compensated for in accordance with the rotary axis positions. If you want to write compensation values with this cycle, the cycle requires **KinematicsComp** (software option 52). With this cycle and using a 3D TS touch probe, you measure a HEIDENHAIN calibration sphere that you have attached to the machine table. The cycle then moves the touch probe automatically to positions in a grid-line arrangement around the calibration sphere. The machine manufacturer defines these swivel axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing operation on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine manufacturer defines this compensation table (*.kco) and specifies its storage location.

When using Cycle 453, run it at different positions in the workspace. This allows you to check immediately if compensation with Cycle 453 has the desired positive effect on the machine's accuracy.

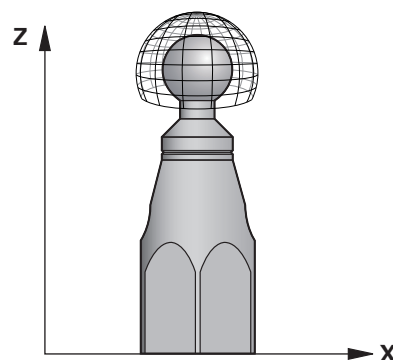
Only when the desired improvements are achieved with the same compensation values at several positions is such a type of compensation suitable for the respective machine. If this is not the case, then the errors are to be sought outside the rotary axes.

Perform the measurement with Cycle 453 in an optimized condition regarding the rotary axis positioning errors. Use, for example, Cycle 451 before doing so.



HEIDENHAIN recommends using the calibration spheres **KKH 250 (order number 655475-01)** or **KKH 100 (order number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The control then optimizes the accuracy of your machine. For this purpose, it automatically saves the compensation values resulting from a measurement in a compensation table (*.kco). (This applies to mode **Q406=1**.)



Cycle run

- 1 Clamp the calibration sphere and check for potential collisions.
- 2 In Manual mode of operation, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
- 3 Select one of the Program Run operating modes and start the NC program
- 4 The cycle is executed in accordance with the setting in **Q406** (-1=Delete mode / 0=Test mode / 1=Compensate mode)

i During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

Various modes (Q406)**Delete mode Q406 = -1 (option 52)**

- The axes are not moved
- The control writes all values to the compensation table (*.kco), setting them to "0". The result is that no further compensations will be effective for the currently selected kinematics.

Test mode Q406 = 0

- The control probes the calibration sphere.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Compensate mode Q406 = 1 (option 52)

- The control probes the calibration sphere.
- The control writes the deviations to the compensation table (*.kco). The table is updated and the compensation settings are immediately effective.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. It is recommended to clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

i Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Notes



KinematicsOpt (software option 48) is required.
KinematicsComp (software option 52) is required.
This function must be enabled and adapted by the machine manufacturer.
Your machine manufacturer defines the storage location of the compensation table (*.kco).

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
 - ▶ Set the preset and the basic rotation again after optimization.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
 - Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
 - As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
 - Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or you set input parameter **Q431** to 1 or 3, respectively.
 - For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
 - Programming in inches: The control always records the log data and results of measurement in millimeters.
 - If you have activated preset setting before the calibration (**Q431** = 1/3), then move the touch probe by the amount of the set-up clearance (**Q320** + **SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.



- If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.

Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the maximum permissible modification of a transformation. If the value is not equal to -1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0° .
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	<p>Q406 Mode (-1/0/+1)</p> <p>Define whether the control will write a value of 0 to the values of the compensation table (*.kco), will check the currently existing deviations, or will perform a compensation. A log file (*.html) is created.</p> <p>-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). (option 52)</p> <p>0: Check TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions but does not write values to the compensation table (*.kco). The control displays the standard and maximum deviation in a log (*.html).</p> <p>1: Compensate TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions and writes the deviations to the compensation table (*.kco). The compensations are then immediately effective. The control displays the standard and maximum deviation in a log (*.html). (option 52)</p> <p>Input: -1, 0, +1</p>
	<p>Q407 Radius of calib. sphere?</p> <p>Enter the exact radius of the calibration sphere being used.</p> <p>Input: 0.0001...99.9999</p>
	<p>Q320 Set-up clearance?</p> <p>Additional distance between touch point and ball tip. Q320 is active in addition to the SET_UP column in the touch probe table. This value has an incremental effect.</p> <p>Input: 0...99999.9999 or PREDEF</p>
	<p>Q408 Retraction height?</p> <p>0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C.</p> <p>> 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253. This value has an absolute effect.</p> <p>Input: 0...99999.9999</p>
	<p>Q253 Feed rate for pre-positioning?</p> <p>Define the traversing speed of the tool during pre-positioning in mm/min.</p> <p>Input: 0...99999.9999 or FMAX, FAUTO, PREDEF</p>

Help graphic

Parameter

Q380 Ref. angle in ref. axis?

Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect.

Input: **0...360**

Q423 Number of probes?

Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision.

Input: **3...8**

Q431 Preset (0/1/2/3)?

Define whether the control will automatically set the active preset at the center of the sphere:

0: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle

1: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

2: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle

3: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

Input: **0, 1, 2, 3**

Probing with Cycle 453

11 TCH PROBE 453 KINEMATICS GRID ~	
Q406=+0	;MODE ~
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+750	;F PRE-POSITIONING ~
Q380=+0	;REFERENCE ANGLE ~
Q423=+4	;NO. OF PROBE POINTS ~
Q431=+0	;PRESET

Log function

After running Cycle **453**, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder where the current NC program resides. It contains the following data:

- Date and time of log 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

9

**Touch Probe Cycles:
Automatic Tool
Measurement**

9.1 Fundamentals

Overview



Refer to your machine manual.

Some cycles and functions may not be provided on your machine.

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













Operating notes

- When running touch probe cycles, Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING** must not be active
- HEIDENHAIN only guarantees the proper operation of the probing cycles if HEIDENHAIN touch probes are used.

In conjunction with the control's tool measurement cycles, the tool touch probe enables you to measure tools automatically: the compensation values for tool length and radius are stored in the tool table and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Measurement of a stationary tool
- Measurement of a rotating tool
- Measurement of individual teeth

You can program the cycles for tool measurement in the **Programming** operating mode using the **TOUCH PROBE** key. The following cycles are available:

New format	Old format	Cycle	Page
		Cycle 30 or 480 CALIBRATE TT <ul style="list-style-type: none"> ■ Calibrating the tool touch probe 	385
		Cycle 31 or 481 CAL. TOOL LENGTH <ul style="list-style-type: none"> ■ Measuring the tool length 	388
		Cycle 32 or 482 CAL. TOOL RADIUS <ul style="list-style-type: none"> ■ Measuring the tool radius 	392
		Cycle 33 or 483 MEASURE TOOL <ul style="list-style-type: none"> ■ Measuring the tool length and radius 	397
		Cycle 484 CALIBRATE IR TT <ul style="list-style-type: none"> ■ Calibrating the tool touch probe (e.g., infrared tool touch probe) 	402
		Cycle 485 MEASURE LATHE TOOL (option 50 or 158) <ul style="list-style-type: none"> ■ Measurement of turning tools 	406



Operating notes:

- The touch probe cycles can be used only when the central tool file TOOL.T is active.
- Before working with the touch probe cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.

Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only the following differences between Cycles **31** to **33** and Cycles **481** to **483**:

- Cycles **481** to **483** are also available as **G481** to **G483** for ISO programming
- Instead of a selectable parameter for the status of the measurement, Cycles **481** to **483** use the fixed parameter **Q199**.

Measuring a tool of length 0



Refer to your machine manual!

The optional machine parameter **maxToolLengthTT** (no. 122607) enables the machine manufacturer to define a maximum tool length for the tool measurement cycles.



HEIDENHAIN recommends that you always define tools with their actual tool length if possible.

The tool measuring cycles measure tools automatically. You can also measure tools defined with a length **L** of 0 in the tool table. To do this, the machine manufacturer must define a maximum tool length value in the optional machine parameter **maxToolLengthTT** (no. 122607). The control starts a search in which the actual tool length is roughly determined in the first step. This is followed by a fine measurement.

Cycle run

- 1 The tool travels to a clearance height centered above the touch probe.
The clearance height equals the value of the optional machine parameter **maxToolLengthTT** (no. 122607).
- 2 The control performs a rough measurement with the spindle standing still.
When measuring a stationary tool, the control will use the feed rate for probing defined in the machine parameter **probingFeed** (no. 122709).
- 3 The control saves the roughly measured length.
- 4 The control performs a fine measurement with the values from the tool measuring cycle.

Notes

NOTICE

Risk of collision!

If the machine manufacturer fails to define the optional machine parameter **maxToolLengthTT** (no. 122607), there will be no tool search. The control pre-positions the tool with a length of 0. Risk of collision!

- ▶ Observe the machine parameter value in the machine manual.
- ▶ Define tools with the actual tool length **L**

NOTICE

Risk of collision!

Risk of collision if the tool is longer than the value of the optional machine parameter **maxToolLengthTT** (no. 122607)!

- ▶ Observe the machine parameter value in the machine manual

Setting machine parameters



- The touch probe cycles **480, 481, 482, 483, 484, 485** can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).



Programming and operating notes:

- Before you start working with the touch probe cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300).
- When measuring a stationary tool, the control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

Setting of the spindle speed

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

$$n = \text{maxPeriphSpeedMeas} / (r \cdot 0.0063) \text{ where}$$

Abbreviation	Definition
n	Shaft speed [rpm]
maxPeriphSpeedMeas	Maximum permissible cutting speed in m/min
r	Active tool radius [mm]

Setting of the feed rate

The probing feed rate is calculated as follows:

$$v = \text{measuring tolerance} \cdot n$$

Abbreviation	Definition
v	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on maxPeriphSpeedMeas
n	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate. The control provides the following options:

- **ConstantTolerance**
- **VariableTolerance**
- **ConstantFeed**

ConstantTolerance:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance (**measureTolerance1** (no. 122715), the sooner you will encounter this effect.

- **VariableTolerance:**

VariableTolerance:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	measureTolerance1
30 to 60 mm	2 • measureTolerance1
60 to 90 mm	3 • measureTolerance1
90 to 120 mm	4 • measureTolerance1

ConstantFeed:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance = $(r \cdot \text{measureTolerance1}) / 5 \text{ mm}$ where

Abbreviation	Definition
r	Active tool radius [mm]
measureTolerance1	Maximum permissible error of measurement

Setting for consideration of parallel axes and changes in the kinematics

Refer to your machine manual.

Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

Auxiliary or parallel axes cannot be probed, regardless of the setting of the optional machine parameter **calPosType** (no. 122606).

If the machine manufacturer changes the setting of the optional machine parameter, you need to recalibrate the tool touch probe.

Entries in the tool table for milling and turning tools

Abbr.	Inputs	Dialog
CUT	The number of teeth of the tool for automatic tool measurement or cutting data calculation (maximum of 20 teeth)	Number of teeth?
LTOL	Permitted tool length deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...5.0000	Wear tolerance: length?
RTOL	Permitted tool radius deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...5.0000	Wear tolerance: radius?
DIRECT.	Cutting direction of the tool for automatic tool measurement with a rotating tool. Input: -, +	Cutting direction (M3 = -)?
R-OFFS	Position of tool upon length measurement, offset between the probe contact center and the tool center for automatic tool measurement. Default setting: No value entered (offset = tool radius) Input: -99999.9999...+99999.9999	Tool offset: radius?
L-OFFS	Position of tool upon radius measurement, distance between the probe contact top edge and the tool tip for automatic tool measurement. Is added to the offsetToolAxis machine parameter (no. 122707). Input: -99999.9999...+99999.9999	Tool offset: length?
LBREAK	Permitted tool length deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...9.0000	Breakage tolerance: length?
RBREAK	Permitted tool radius deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...9.0000	Breakage tolerance: radius?

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	No function	0: No offset required because tool tip is to be measured	
End mill	4: four cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from offsetToolAxis (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: four cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

9.2 Cycle 30 or 480 CALIBRATE TT

ISO programming

G480

Application



Refer to your machine manual!

You calibrate the TT with touch probe cycle **30** or **480** (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 379). The calibration process runs automatically. The control also measures the center offset of the calibration tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

You calibrate the TT with touch probe cycle **30** or **480**.

Touch probe

For the touch probe you use a spherical or cuboid probe contact

Cuboid probe contact

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined. Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

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



When clamping the tool touch probe, make sure that the edges of the cuboid probe contact are aligned as parallel to the machine axes as possible. The angle of misalignment should be less than 1° and the tilt angle should be less than 0.3°.

Calibration tool

The calibration tool must be a precisely cylindrical part, for example a cylindrical pin. The resulting calibration values are stored in the control memory and are accounted for during subsequent tool measurement.

Cycle run

- 1 Clamp the calibration tool. The calibration tool must be a precisely cylindrical part, for example a cylindrical pin
- 2 Manually position the calibration tool in the working plane over the center of the TT
- 3 Position the calibration tool in the tool axis at approximately 15 mm plus set-up clearance over the TT
- 4 The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
- 5 The calibration process along the tool axis starts
- 6 This is followed by calibration in the working plane
- 7 The control positions the calibration tool in the working plane at a position of TT radius + set-up clearance + 11 mm
- 8 Then the control moves the tool downwards along the tool axis and the calibration process starts
- 9 During probing, the control moves in a square pattern
- 10 The control saves the calibration values and considers them during subsequent tool measurement
- 11 The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.

Notes about machine parameters

- Use the machine parameter **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300) to define the functionality of the calibration cycle. Refer to your machine manual.
 - Use the machine parameter **centerPos** to define the position of the TT within the machine's working space.
- The TT needs to be recalibrated if you change the position of the TT on the table and/or a **centerPos** machine parameter.
- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the calibration tool above the top of the probe contact (safety zone from safetyDistToolAx (no. 114203)).</p> <p>Input: -99999.9999...+99999.9999</p>

ExampleNew format

```
11 TOOL CALL 12 Z
12 TCH PROBE 480 CALIBRATE TT ~
Q260=+100 ;CLEARANCE HEIGHT
```

Example of old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 30.0 CALIBRATE TT
13 TCH PROBE 30.1 HEIGHT:+90
```

9.3 Cycle 31 or 481 CAL. TOOL LENGTH

ISO programming

G481

Application



Refer to your machine manual!

For measuring the tool length, program touch probe cycle **31** or **482** (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 379). 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, set the parameter **PROBING THE TEETH** = 1 in Cycle **31**.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the **TOOL.T** tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- You can run an individual tooth measurement for tools with **up to 20 teeth**.
- Cycles **31** and **481** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**LBREAK** and **LTOL**) from the **TOOL.T** table.

Q340: 0 and 1

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools. **Further information:** User's Manual for Setup, Testing and Running NC Programs

Cycle parameters

Help graphic

Parameter

Q340 Tool measurement mode (0-2)?

Define whether and how the measured data will be entered in the tool table.

0: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL = 0. If there is already a value in TOOL.T, it will be overwritten.

1: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation is also available in the Q parameter **Q115**. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T).

2: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter **Q115**. Nothing is entered under L or DL in the tool table.

Input: **0, 1, 2**



Note the behavior with grinding tools,

Further information: "Measuring grinding tools",
Page 389

Q260 Clearance height?

Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**).

Input: **-99999.9999...+99999.9999**

Q341 Probe the teeth? 0=no/1=yes

Define whether the control will measure the individual teeth (maximum of 20 teeth)

Input: **0, 1**

Example new format

```
11 TOOL CALL 12 Z
```

```
12 TCH PROBE 481 CAL. TOOL LENGTH ~
```

```
Q340=+1 ;CHECK ~
```

```
Q260=+100 ;CLEARANCE HEIGHT ~
```

```
Q341=+1 ;PROBING THE TEETH
```

Cycle **31** includes an additional parameter:

Help graphic	Parameter
	<p>Parameter number for result?</p> <p>Parameter number in which the control stores the status of the measurement:</p> <p>0.0: Tool is within the tolerance</p> <p>1.0: Tool is worn (LTOL exceeded)</p> <p>2.0: Tool is broken (LBREAK exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with NO ENT</p> <p>Input: 0...1999</p>

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:0
14 TCH PROBE 31.2 HEIGHT::+120
15 TCH PROBE 31.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:1 Q5
14 TCH PROBE 31.2 HEIGHT:+120
15 TCH PROBE 31.3 PROBING THE TEETH:1
```

9.4 Cycle 32 or 482 CAL. TOOL RADIUS

ISO programming

G482

Application



Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **32** or **482** (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 379). Select via input parameters by which of two methods the tool radius is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). The control probes the tool radially while it is rotating.

If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Further information: "Notes for individual tooth measurement Q341=1", Page 394

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- Cycles **32** and **482** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**RBREAK** and **RTOL**) from the **TOOL.T** table.

Q340=0 or 1

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools. **Further information:** User's Manual for Setup, Testing and Running NC Programs

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement Q341=1**NOTICE****Caution: Danger to the tool and workpiece!**

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
- ▶ Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

$$\varepsilon = 90 - \operatorname{atan} \left(\frac{h_{[tt]}}{R \times 2 \times \pi \times x} \right)$$

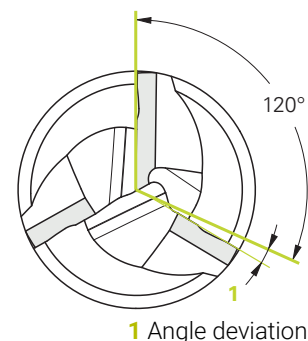
Abbreviation	Definition
ε	Maximum angle of twist
$h_{[tt]}$	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool

- i** On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist. Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

NOTICE**Caution: Possible material damage!**

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.

- ▶ Check the workpiece dimensions during subsequent machining operations



Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.

Cycle parameters

Help graphic	Parameter
	<p>Q340 Tool measurement mode (0-2)?</p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p>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.</p> <p>1: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation is also available in the Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T).</p> <p>2: The measured tool radius is compared to the tool radius from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q116. Nothing is entered under R or DR in the tool table.</p> <p>Input: 0, 1, 2</p>
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus).</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q341 Probe the teeth? 0=no/1=yes</p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: 0, 1</p>

Example new format

11 TOOL CALL 12 Z
12 TCH PROBE 482 CAL. TOOL RADIUS ~
Q340=+1 ;CHECK ~
Q260=+100 ;CLEARANCE HEIGHT ~
Q341=+1 ;PROBING THE TEETH

Cycle **32** includes an additional parameter:

Help graphic	Parameter
	<p>Parameter number for result?</p> <p>Parameter number in which the control stores the status of the measurement:</p> <p>0.0: Tool is within the tolerance</p> <p>1.0: Tool is worn (RTOL exceeded)</p> <p>2.0: Tool is broken (RBREAK exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with NO ENT</p> <p>Input: 0...1999</p>

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:0
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:1 Q5
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:1
```

9.5 Cycle 33 or 483 MEASURE TOOL

ISO programming

G483

Application



Refer to your machine manual!

To measure the tool completely (length and radius), program touch probe cycle **33** or **483** (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 379). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Input parameters allow you to select which of the two following methods will be used to measure the tool:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

Measuring the tool while it is rotating:

The control measures the tool in a fixed programmed sequence. First, if possible, it measures the tool length, and then the tool radius.

Measuring the individual teeth:

The control measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for touch probe cycles **31** and **32** as well as **481** and **482**.

Further information: "Notes for individual tooth measurement of radius Q341=1", Page 399

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the **TOOL.T** tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- Cycles **33** and **483** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**LBREAK**, **RBREAK**, **LTOL**, and **RTOL**) from the **TOOL.T** table.

Q340: 0 and 1

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools. **Further information:** User's Manual for Setup, Testing and Running NC Programs

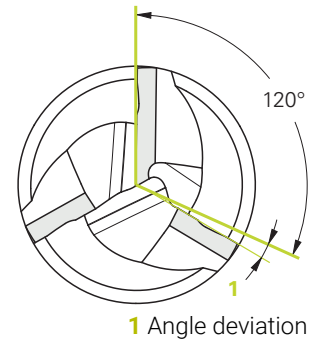
Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement of radius Q341=1**NOTICE****Caution: Danger to the tool and workpiece!**

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
- ▶ Check the workpiece optically in order to exclude broken tools



If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

$$\varepsilon = 90 - \operatorname{atan} \left(\frac{h_{[tt]}}{\frac{R \times 2 \times \pi}{x}} \right)$$

Abbreviation	Definition
ε	Maximum angle of twist
$h_{[tt]}$	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool

i On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist. Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

NOTICE**Caution: Possible material damage!**

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.

- ▶ Check the workpiece dimensions during subsequent machining operations

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.

Cycle parameters

Help graphic	Parameter
	<p>Q340 Tool measurement mode (0-2)?</p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p>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.</p> <p>1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in the Q parameters Q115 and Q116. If the delta value is greater than the permissible tool length or tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T).</p> <p>2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored values and writes it to the Q parameter Q115 or Q116. Nothing is entered under L, R, or DL, DR in the tool table.</p> <p>Input: 0, 1, 2</p>
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus).</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q341 Probe the teeth? 0=no/1=yes</p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: 0, 1</p>

Example new format

11 TOOL CALL 12 Z
12 TCH PROBE 483 MEASURE TOOL ~
Q340=+1 ;CHECK ~
Q260=+100 ;CLEARANCE HEIGHT ~
Q341=+1 ;PROBING THE TEETH

Cycle **33** includes an additional parameter:

Help graphic	Parameter
	<p>Parameter number for result?</p> <p>Parameter number in which the control stores the status of the measurement:</p> <p>0.0: Tool is within the tolerance</p> <p>1.0: Tool is worn (LTOL or/and RTOL exceeded)</p> <p>2.0: Tool is broken (LBREAK or/and RBREAK exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with NO ENT.</p> <p>Input: 0...1999</p>

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:0
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:1 Q5
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:1
```

9.6 Cycle 484 CALIBRATE IR TT

ISO programming

G484

Application

Cycle **484** allows you to calibrate your tool touch probe (e.g., the wireless infrared TT 460 tool touch probe). You can perform the calibration process with or without manual intervention.

- **With manual intervention:** If you define **Q536** = 0, then the control will stop before the calibration process. You then need to position the calibration tool manually above the center of the tool touch probe.
- **Without manual intervention:** If you define **Q536** = 1, then the control will automatically execute the cycle. You may have to program a repositioning movement before. This depends on the value of the parameter **Q523 POSITION TT**.

Cycle run



Refer to your machine manual.

The machine manufacturer defines the functionality of the cycle.

To calibrate the tool touch probe, program the touch probe cycle **484**. In input parameter **Q536**, you can specify whether you want to run the cycle with or without manual intervention.

Touch probe

For the touch probe you use a spherical or cuboid probe contact

Cuboid probe contact:

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined. Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

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



When clamping the tool touch probe, make sure that the edges of the cuboid probe contact are aligned as parallel to the machine axes as possible. The angle of misalignment should be less than 1° and the tilt angle should be less than 0.3°.

Calibration tool:

The calibration tool must be a precisely cylindrical part, for example a cylindrical pin. Enter the exact length and radius of the calibration tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Q536 = 0: With manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibration tool
- ▶ Start the calibration cycle
- > The control interrupts the calibration cycle and displays a dialog in a new window.
- ▶ Manually position the calibration tool above the center of the tool touch probe.



Ensure that the calibration tool is located above the measuring surface of the probe contact.

- ▶ Press **NC Start** to resume cycle run
- > If you have programmed **Q523 = 2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

Q536 = 1: Without manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibrating tool
- ▶ Position the calibration tool above the center of the tool touch probe before the start of the cycle.



- Ensure that the calibration tool is located above the measuring surface of the probe contact.
- For a calibration process without manual intervention, you do not need to position the calibration tool above the center of the tool touch probe. The cycle adopts the position from the machine parameters and automatically moves the tool to this position.

- ▶ Start the calibration cycle
- > The calibration cycle is executed without stopping.
- > If you have programmed **Q523 = 2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

Notes

NOTICE

Danger of collision!

If you program **Q536=1**, the tool must be pre-positioned before calling the cycle. The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle. There is a danger of collision!

- ▶ Specify whether to stop before cycle start or run the cycle automatically without stopping.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. If you use a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. Major inaccuracies may occur if you use a calibration tool whose diameter is too small and/or that protrudes too far from the chuck.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.
- The TT needs to be recalibrated if you change its position on the table.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	<p>Q536 Stop before running (0=Stop)?</p> <p>Define whether the control will stop before the calibration process or whether the cycle will automatically be executed without a stop:</p> <p>0: Stop before the calibration process. The control prompts you to position the calibration tool manually above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC Start to continue the calibration process or press the CANCEL soft key to cancel the calibration process.</p> <p>1: Without stopping before the calibration process. The control starts the calibration process depending on Q523. Before running Cycle 484, you may have to position the tool above the tool touch probe.</p> <p>Input: 0, 1</p>
	<p>Q523 Position of tool probe (0-2)?</p> <p>Position of the tool touch probe:</p> <p>0: Current position of the calibration tool. The tool touch probe is below the current position of the calibration tool. If Q536 = 0, position the calibration tool manually above the center of the tool touch probe during the cycle. If Q536 = 1, you need to position the calibration tool above the center of the tool touch probe before the start of the cycle.</p> <p>1: Configured position of the tool touch probe. The control adopts the position from the machine parameter centerPos (no. 114201). You do not need to pre-position the tool. The calibration tool approaches the position automatically.</p> <p>2: Current position of the calibration tool. See Q523 = 0. 0. The control additionally writes the determined position (where applicable) to the machine parameter centerPos (no. 114201) after calibration.</p> <p>Input: 0, 1, 2</p>

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 484 CALIBRATE IR TT ~	
Q536=+0	;STOP BEFORE RUNNING ~
Q523=+0	;TT POSITION

9.7 Cycle 485 MEASURE LATHE TOOL (option 50 or 158)

ISO programming

G485

Application



Refer to your machine manual!

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

Cycle **485 MEASURE LATHE TOOL** is available for the measurement of turning tools with a tool touch probe from HEIDENHAIN. The control measures the tool in a fixed programmed sequence.

Cycle run

- 1 The control positions the turning tool to the clearance height
- 2 The turning tool is oriented based on the entries in **TO** and **ORI**
- 3 The control moves the tool to the measuring position in the main axis; the traverse movement is interpolated in the main and secondary axes
- 4 Then the turning tool moves to the measuring position in the tool axis
- 5 The tool is measured. Depending on the definition of **Q340**, either tool dimensions are changed or the tool is locked
- 6 The measuring result is transferred to the result parameter **Q199**
- 7 After the measurement has been performed, the control positions the tool in the tool axis to the clearance height

Result parameter Q199:

Result	Meaning
0	Tool dimensions within the tolerance LTOL / RTOL Tool is not locked
1	Tool dimensions outside the tolerance LTOL / RTOL Tool is locked
2	Tool dimensions outside the tolerance LBREAK / RBREAK Tool is locked

The cycle uses the following entries from toolturn.trn:

Abbr.	Entries	Dialog
ZL	Tool length 1 (Z direction)	Tool length 1?
XL	Tool length 2 (X direction)	Tool length 2?
DZL	Delta value of tool length 1 (Z direction), is added to ZL	Tool length oversize 1
DXL	Delta value of tool length 2 (X direction), is added to XL	Tool length oversize 2
RS	Cutting edge radius: If contours were programmed with radius compensation RL or RR , the control takes the cutter radius into account in turning cycles, and performs tool tip radius compensation	Cutting edge radius?
TO	Tool orientation: From the tool orientation, the control determines the position of the tool cutting edge and, depending on the selected tool type, additional information such as the tool angle direction, position of the tool reference point, etc. This information is necessary, for example, for calculating the tool tip radius compensation, milling cutter radius compensation, plunge angle, etc.	Tool orientation?
ORI	Spindle orientation angle: Angle of the indexable insert to the main axis	Angle of spindle orientation?
TYPE	Type of turning tool: Roughing tool ROUGH , finishing tool FINISH , threading tool THREAD , recessing tool RECESS , button tool BUTTON , recess-turning tool RECTURN	Type of turning tool

Further information: "Tool orientation (TO) that is supported for the following types of turning tools (TYPE)", Page 408

Tool orientation (TO) that is supported for the following types of turning tools (TYPE)

TYPE	Supported TO with possible limitations	Non-supported TO	
ROUGH, FINISH	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 2, only XL ■ 3, only XL ■ 5, only XL ■ 6, only XL ■ 8, only ZL ■ 18 	<ul style="list-style-type: none"> ■ 4 ■ 9 	
BUTTON	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 2, only XL ■ 3, only XL ■ 5, only XL ■ 6, only XL ■ 8, only ZL 	<ul style="list-style-type: none"> ■ 4 ■ 9 	
RECESS, RECTURN	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only XL ■ 5, only XL 	<ul style="list-style-type: none"> ■ 4 ■ 6 ■ 9 	
THREAD	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only XL ■ 5, only XL 	<ul style="list-style-type: none"> ■ 4 ■ 6 ■ 9 	

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

NOTICE

Danger of collision!

If the tool data **ZL / DZL** and **XL / DXL** deviate by more than ± 2 mm from the real tool data, then there is a danger of collision.

- ▶ Enter the approximate tool data closer than ± 2 mm
- ▶ Run the cycle carefully

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before you begin the cycle, you must run a **TOOL CALL** with the tool axis **Z**.
- If you define **YL** and **DYL** with a value outside of ± 5 mm, the tool won't reach the tool touch probe.
- The cycle does not support **SPB-INSERT** (angular offset). You must enter the value 0 in **SPB-INSERT**, otherwise the control will generate an error message.

Note regarding machine parameters

- The cycle depends on the optional machine parameter **CfgTTRectStylus** (no. 114300). Refer to your machine manual.

Cycle parameters

Help graphic	Parameter
	<p>Q340 Tool measurement mode (0-2)?</p> <p>Use of the measured values:</p> <p>0: The measured values are entered in ZL and XL. If values are already entered in the tool table, they will be overwritten. DZL and DXL will be reset to 0. TL will not be changed</p> <p>1: The measured values ZL and XL are compared with the values from the tool table. These values will not be changed. The control then calculates the deviations of ZL and XL, and enters these in DZL and DXL. If the delta values are larger than the permissible wear or breakage tolerance, the control locks the tool (TL = Tool Locked). In addition, the deviation is also entered in the Q parameters Q115 and Q116</p> <p>2: The measured values ZL and XL as well as DZL and DXL are compared with the values from the tool table, but are not changed. If the values are larger than the permissible wear or breakage tolerance, the control locks the tool (TL = Tool Locked).</p> <p>Input: 0, 1, 2</p>
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus).</p> <p>Input: -99999.9999...+99999.9999</p>

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 485 MEASURE LATHE TOOL ~	
Q340=+1	;CHECK ~
Q260=+100	;CLEARANCE HEIGHT



10











**Cycles: Special
Functions**





10.1 Fundamentals

Overview

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

-  ▶ Press the **CYCL DEF** key
-  ▶ Press the **SPECIAL CYCLES** soft key

Soft key	Cycle	Page
	9 DWELL TIME <ul style="list-style-type: none"> ■ Delay execution by the programmed dwell time 	Further information: User's Manual for Programming of Machining Cycles
	12 PGM CALL <ul style="list-style-type: none"> ■ Call any NC program 	Further information: User's Manual for Programming of Machining Cycles
	13 ORIENTATION <ul style="list-style-type: none"> ■ Rotate spindle to a specific angle 	414
	32 TOLERANCE <ul style="list-style-type: none"> ■ Program the permissible contour deviation for jerk-free machining operations 	Further information: User's Manual for Programming of Machining Cycles
	291 COUPLG.TURNG.INTERP. <ul style="list-style-type: none"> ■ Coupling of the tool spindle with the positions of the linear axes ■ Or, rescind the spindle coupling 	Further information: User's Manual for Programming of Machining Cycles
	292 CONTOUR.TURNG.INTRP. <ul style="list-style-type: none"> ■ Coupling of the tool spindle with the positions of the linear axes ■ Create certain rotationally symmetric contours in the active working plane ■ Possible with tilted machining plane 	Further information: User's Manual for Programming of Machining Cycles
	225 ENGRAVING <ul style="list-style-type: none"> ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc 	Further information: User's Manual for Programming of Machining Cycles
	232 FACE MILLING <ul style="list-style-type: none"> ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan 	Further information: User's Manual for Programming of Machining Cycles
	285 DEFINE GEAR <ul style="list-style-type: none"> ■ Define the geometry of the gear wheel 	Further information: User's Manual for Programming of Machining Cycles
	286 GEAR HOBBING <ul style="list-style-type: none"> ■ Definition of the tool data ■ Selection of the machining strategy and side ■ Possibility of using the entire cutting edge 	Further information: User's Manual for Programming of Machining Cycles

Soft key	Cycle	Page
	287 GEAR SKIVING <ul style="list-style-type: none"> ■ Definition of the tool data ■ Selection of the machining side ■ Definition of the first and last infeed ■ Definition of the number of cuts 	Further information: User's Manual for Programming of Machining Cycles
	238 MEASURE MACHINE STATUS <ul style="list-style-type: none"> ■ Determine the current machine status or test the measuring sequence 	Further information: User's Manual for Programming of Machining Cycles
	239 ASCERTAIN THE LOAD <ul style="list-style-type: none"> ■ Selection for a weighing run ■ Reset the load-dependent feedforward and controller parameters 	Further information: User's Manual for Programming of Machining Cycles
	18 THREAD CUTTING <ul style="list-style-type: none"> ■ With controlled spindle ■ Spindle stops at the bottom of the hole 	Further information: User's Manual for Programming of Machining Cycles

10.2 Cycle 13 ORIENTATION

ISO programming

G36

Application



Refer to your machine manual.

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

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

Oriented spindle stops are required for purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the transceiver window of HEIDENHAIN 3D touch probes with infrared transmission

With **M19** or **M20**, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program **M19** or **M20** without having defined Cycle **13** beforehand, the control positions the main spindle at an angle that has been set by the machine manufacturer.

Notes

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

Cycle parameters

Help graphic

Parameter

Orientation angle

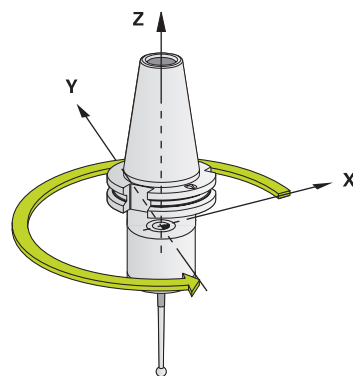
Enter the angle relative to the angle reference axis of the working plane.

Input: **0...360**

Example

```
11 CYCL DEF 13.0 ORIENTATION
```

```
12 CYCL DEF 13.1 ANGLE180
```



11

Tables of Cycles

11.1 Table of cycles



All cycles that are not related to the measuring cycles are described in the **Programming of Machining Cycles** User's Manual. This manual is available from HEIDENHAIN upon request.

ID of User's Manual for Programming of Machining Cycles:
1303406-xx

Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	REF. PLANE	■		240
1	POLAR PRESET	■		242
3	MEASURING	■		295
4	MEASURING IN 3-D	■		298
30	CALIBRATE TT	■		385
31	CAL. TOOL LENGTH	■		388
32	CAL. TOOL RADIUS	■		392
33	MEASURE TOOL	■		397
400	BASIC ROTATION	■		105
401	ROT OF 2 HOLES	■		109
402	ROT OF 2 STUDS	■		114
403	ROT IN ROTARY AXIS	■		119
404	SET BASIC ROTATION	■		128
405	ROT IN C AXIS	■		124
408	SLOT CENTER PRESET	■		221
409	RIDGE CENTER PRESET	■		226
410	PRESET INSIDE RECTAN	■		167
411	PRESET OUTS. RECTAN	■		172
412	PRESET INSIDE CIRCLE	■		178
413	PRESET OUTS. CIRCLE	■		184
414	PRESET OUTS. CORNER	■		190
415	PRESET INSIDE CORNER	■		196
416	PRESET CIRCLE CENTER	■		202
417	PRESET IN TS AXIS	■		208
418	PRESET FROM 4 HOLES	■		212
419	PRESET IN ONE AXIS	■		217
420	MEASURE ANGLE	■		244
421	MEASURE HOLE	■		247
422	MEAS. CIRCLE OUTSIDE	■		253
423	MEAS. RECTAN. INSIDE	■		259

Cycle number	Cycle name	DEF active	CALL active	Page
424	MEAS. RECTAN. OUTS.	■		264
425	MEASURE INSIDE WIDTH	■		268
426	MEASURE RIDGE WIDTH	■		272
427	MEASURE COORDINATE	■		276
430	MEAS. BOLT HOLE CIRC	■		281
431	MEASURE PLANE	■		286
441	FAST PROBING	■		306
444	PROBING IN 3-D	■		300
450	SAVE KINEMATICS	■		336
451	MEASURE KINEMATICS	■		339
452	PRESET COMPENSATION	■		357
453	KINEMATICS GRID	■		370
460	CALIBRATION OF TS ON A SPHERE	■		324
461	TS CALIBRATION OF TOOL LENGTH	■		316
462	CALIBRATION OF A TS IN A RING	■		318
463	TS CALIBRATION ON STUD	■		321
480	CALIBRATE TT	■		385
481	CAL. TOOL LENGTH	■		388
482	CAL. TOOL RADIUS	■		392
483	MEASURE TOOL	■		397
484	CALIBRATE IR TT	■		402
485	MEASURE LATHE TOOL	■		406
620	VT 121 CALIBRATION	■		Further information: VTC User's Manual ID: 1322445-xx
621	MANUAL INSPECTION	■		Further information: VTC User's Manual ID: 1322445-xx
622	IMAGES	■		Further information: VTC User's Manual ID: 1322445-xx
623	BREAKAGE CHECK	■		Further information: VTC User's Manual ID: 1322445-xx
624	MEASUREMENT OF CUTTING EDGE ANGLE	■		Further information: VTC User's Manual ID: 1322445-xx
1400	POSITION PROBING	■		135
1401	CIRCLE PROBING	■		139
1402	SPHERE PROBING	■		144

Cycle number	Cycle name	DEF active	CALL active	Page
1404	PROBE SLOT/RIDGE	■		149
1410	PROBING ON EDGE	■		71
1411	PROBING TWO CIRCLES	■		79
1412	INCLINED EDGE PROBING	■		87
1416	INTERSECTION PROBING	■		95
1420	PROBING IN PLANE	■		64
1430	PROBE POSITION OF UNDERCUT	■		153
1434	PROBE SLOT/RIDGE UNDERCUT	■		159
1493	EXTRUSION PROBING	■		309

Machining cycles

Cycle number	Cycle name	DEF active	CALL active	Page
13	ORIENTATION	■		414

Index

3

3D Touch Probes..... 38

A

About this manual..... 20
 Automatic preset setting
 Bolt hole circle..... 202
 Center of 4 holes..... 212
 Circle..... 139
 Circular pocket (hole)..... 178
 Circular stud..... 184
 Fundamentals of 14xx..... 134
 Fundamentals of 4xx..... 165
 Inside corner..... 196
 Outside corner..... 190
 Rectangular pocket..... 167
 Rectangular stud..... 172
 Reference plane..... 240
 Ridge..... 149
 Ridge center..... 226
 Ridge undercut..... 159
 Single axis..... 217
 Single position..... 135
 Slot..... 149
 Slot center..... 221
 Slot undercut..... 159
 Sphere..... 144
 Touch probe axis..... 208
 Undercut position..... 153

B

Basic rotation..... 105
 Over two holes..... 109
 over two studs..... 114, 119
 Setting directly..... 128

C

Calibration
 L probe..... 324
 Normal probe..... 324
 Calibration cycles..... 314
 TS calibration in ring..... 318
 TS calibration on sphere..... 324
 TS calibration on stud..... 321
 TS length..... 316
 Calibration of tool touch probe
 Calibration of IR TT..... 402
 Calibration of TT..... 385
 Classification of results..... 237

D

Determine inclined workpiece position
 Touch probe cycles 14xx fundamentals..... 53
 Determining workpiece

misalignment
 Basic rotation..... 105
 Basic rotation over two holes 109
 Basic rotation over two studs..... 114, 119
 Fundamentals of touch probe cycles 400-405..... 104
 Inclined edge probing..... 87
 Intersection probing..... 95
 Probing in plane..... 64
 Probing on edge..... 71
 Probing two circles..... 79
 Rotation around C axis..... 124
 Setting the basic rotation..... 128

E

Extrusion probing..... 309

F

Fast probing..... 306
 Feature content level..... 27

G

GLOBAL DEF..... 46

I

Inclined workpiece position checking
 Fundamentals..... 234

K

Kinematics measurement
 Accuracy..... 345
 Backlash..... 347
 Fundamentals..... 332
 Hirth coupling..... 343
 Kinematics grid..... 370
 Measure kinematics..... 339
 Prerequisites..... 334
 Preset compensation..... 357
 Saving kinematics..... 336
 KinematicsOpt..... 332

M

Measuring
 Angle..... 244
 Bolt hole circle..... 281
 Circle outside..... 253
 Coordinate..... 276
 Hole..... 247
 Inside width..... 268
 Plane..... 286
 Rectangle inside..... 259
 Rectangle outside..... 264
 Ridge width..... 272
 Measuring in 3D..... 298
 Measuring with Cycle 3..... 295

O

Option..... 23

P

Positioning logic..... 45
 Probing feed rate..... 43
 Probing in 3D..... 300

R

Recording measurement results.... 235

S

Software option..... 23
 Spindle orientation..... 414

T

Table of cycles..... 416
 Touch probe cycles..... 416
 Tolerance monitoring..... 237
 Tool compensation..... 238
 Tool measurement
 Complete measurement..... 397
 Fundamentals..... 378
 Lathe tool measurement..... 406
 Machine parameters..... 381
 Tool length..... 388
 Tool radius..... 392
 Tool table..... 383
 Tool monitoring..... 238
 Touch probe cycles 14xx
 Circle probing..... 139
 Evaluation of tolerances..... 60
 Fundamentals..... 53
 Inclined edge probing..... 87
 Intersection probing..... 95
 Position probing..... 135
 Probing in plane..... 64
 Probing on edge..... 71
 Probing two circles..... 79
 Ridge probing..... 149
 Ridge undercut probing..... 159
 Semi-automatic mode..... 55
 Slot probing..... 149
 Slot undercut probing..... 159
 Sphere probing..... 144
 Transfer an actual position..... 63
 Undercut position probing..... 153

W

Workpiece, checking automatically
 Polar preset..... 242

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104
service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101
service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103
service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102
service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com

The Information Site for
HEIDENHAIN Controls

Klartext App

Klartext on your
mobile device

Google
Play Store

Apple
App Store



Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.



For more details on touch probes and vision systems:

www.heidenhain.com/products/touch-probes-and-vision-systems

