



# HEIDENHAIN



## TNC 320

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

NC Software  
77185x-18

English (en)  
10/2023



## Table of contents

<b>1</b>	<b>Fundamentals.....</b>	<b>17</b>
<b>2</b>	<b>Fundamentals / Overviews.....</b>	<b>35</b>
<b>3</b>	<b>Using Touch Probe Cycles.....</b>	<b>39</b>
<b>4</b>	<b>Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....</b>	<b>53</b>
<b>5</b>	<b>Touch Probe Cycles: Automatic Presetting.....</b>	<b>133</b>
<b>6</b>	<b>Touch Probe Cycles: Automatic Workpiece Inspection.....</b>	<b>235</b>
<b>7</b>	<b>Touch Probe Cycles: Special Functions.....</b>	<b>293</b>
<b>8</b>	<b>Touch Probe Cycles: Automatic Tool Measurement.....</b>	<b>325</b>
<b>9</b>	<b>Cycles: Special Functions.....</b>	<b>355</b>
<b>10</b>	<b>Tables of Cycles.....</b>	<b>359</b>



<b>1</b>	<b>Fundamentals.....</b>	<b>17</b>
1.1	About this manual.....	18
1.2	Control model, software, and features.....	20
	Software options.....	21
	New cycle functions of software 77185x-18.....	25
	Modified cycle functions with 77185x-18.....	30

<b>2</b>	<b>Fundamentals / Overviews.....</b>	<b>35</b>
2.1	Introduction.....	36
2.2	Available cycle groups.....	37
	Overview of machining cycles.....	37
	Overview of touch probe cycles.....	38

<b>3</b>	<b>Using Touch Probe Cycles.....</b>	<b>39</b>
<b>3.1</b>	<b>General information about touch probe cycles.....</b>	<b>40</b>
	Method of function.....	40
	Consideration of a basic rotation in the Manual operation mode.....	41
	Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes.....	41
	Touch probe cycles for automatic operation.....	42
<b>3.2</b>	<b>Before you start working with touch probe cycles.....</b>	<b>44</b>
	Maximum traverse to touch point: DIST in touch probe table.....	44
	Set-up clearance to touch point: SET_UP in touch probe table.....	44
	Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table.....	44
	Touch trigger probe, probing feed rate: F in touch probe table.....	45
	Touch trigger probe, rapid traverse for positioning: FMAX.....	45
	Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table.....	45
	Executing touch probe cycles.....	46
<b>3.3</b>	<b>Program defaults for cycles.....</b>	<b>48</b>
	Overview.....	48
	Entering GLOBAL DEF definitions.....	48
	Using GLOBAL DEF information.....	49
	Global data valid everywhere.....	50
	Global data for probing functions.....	51

<b>4</b>	<b>Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....</b>	<b>53</b>
<b>4.1</b>	<b>Overview.....</b>	<b>54</b>
<b>4.2</b>	<b>Touch probe cycles 14xx: fundamentals.....</b>	<b>55</b>
	Characteristics common to touch probe cycles 14xx for measuring rotations.....	55
	Semi-automatic mode.....	57
	Evaluation of tolerances.....	62
	Transferring the actual position.....	65
<b>4.3</b>	<b>Cycle 1420 PROBING IN PLANE.....</b>	<b>66</b>
	Cycle parameters.....	69
<b>4.4</b>	<b>Cycle 1410 PROBING ON EDGE.....</b>	<b>73</b>
	Cycle parameters.....	77
<b>4.5</b>	<b>Cycle 1411 PROBING TWO CIRCLES.....</b>	<b>81</b>
	Cycle parameters.....	85
<b>4.6</b>	<b>Cycle 1412 INCLINED EDGE PROBING.....</b>	<b>89</b>
	Cycle parameters.....	93
<b>4.7</b>	<b>Cycle 1416 INTERSECTION PROBING.....</b>	<b>97</b>
	Cycle parameters.....	101
<b>4.8</b>	<b>Fundamentals of touch probe cycles 400 to 405.....</b>	<b>106</b>
	Characteristics common to all touch probe cycles for measuring workpiece misalignment.....	106
<b>4.9</b>	<b>Cycle 400 BASIC ROTATION.....</b>	<b>107</b>
	Cycle parameters.....	109
<b>4.10</b>	<b>Cycle 401 ROT OF 2 HOLES.....</b>	<b>111</b>
	Cycle parameters.....	113
<b>4.11</b>	<b>Cycle 402 ROT OF 2 STUDS.....</b>	<b>116</b>
	Cycle parameters.....	118
<b>4.12</b>	<b>Cycle 403 ROT IN ROTARY AXIS.....</b>	<b>121</b>
	Cycle parameters.....	123
<b>4.13</b>	<b>Cycle 405 ROT IN C AXIS.....</b>	<b>126</b>
	Cycle parameters.....	128
<b>4.14</b>	<b>Cycle 404 SET BASIC ROTATION.....</b>	<b>130</b>
	Cycle parameters.....	130
<b>4.15</b>	<b>Example: Determining a basic rotation from two holes.....</b>	<b>131</b>



<b>5</b>	<b>Touch Probe Cycles: Automatic Presetting</b>	<b>133</b>
5.1	Overview	134
5.2	Fundamentals of touch probe cycles 14xx for preset setting	136
	Characteristics common to all touch probe cycles 14xx for preset setting	136
5.3	Cycle 1400 POSITION PROBING	137
	Cycle parameters	139
5.4	Cycle 1401 CIRCLE PROBING	141
	Cycle parameters	143
5.5	Cycle 1402 SPHERE PROBING	146
	Cycle parameters	148
5.6	Cycle 1404 PROBE SLOT/RIDGE	151
	Cycle parameters	153
5.7	Cycle 1430 PROBE POSITION OF UNDERCUT	155
	Cycle parameters	158
5.8	Cycle 1434 PROBE SLOT/RIDGE UNDERCUT	161
	Cycle parameters	164
5.9	Fundamentals of touch probe cycles 408 to 419 for preset setting	167
	Application	167
5.10	Cycle 410 PRESET INSIDE RECTAN.	169
	Cycle parameters	171
5.11	Cycle 411 PRESET OUTS. RECTAN.	174
	Cycle parameters	176
5.12	Cycle 412 PRESET INSIDE CIRCLE	180
	Cycle parameters	182
5.13	Cycle 413 PRESET OUTS. CIRCLE	186
	Cycle parameters	188
5.14	Cycle 414 PRESET OUTS. CORNER	192
	Cycle parameters	194
5.15	Cycle 415 PRESET INSIDE CORNER	198
	Cycle parameters	200
5.16	Cycle 416 PRESET CIRCLE CENTER	204
	Cycle parameters	206

<b>5.17 Cycle 417 PRESET IN TS AXIS.....</b>	<b>210</b>
Cycle parameters.....	212
<b>5.18 Cycle 418 PRESET FROM 4 HOLES.....</b>	<b>214</b>
Cycle parameters.....	216
<b>5.19 Cycle 419 PRESET IN ONE AXIS.....</b>	<b>219</b>
Cycle parameters.....	221
<b>5.20 Cycle 408 SLOT CENTER PRESET.....</b>	<b>223</b>
Cycle parameters.....	225
<b>5.21 Cycle 409 RIDGE CENTER PRESET.....</b>	<b>228</b>
Cycle parameters.....	230
<b>5.22 Example: Presetting at center of a circular segment and on top surface of workpiece.....</b>	<b>233</b>
<b>5.23 Example: Presetting on top surface of workpiece and at center of a bolt hole circle.....</b>	<b>234</b>

<b>6</b>	<b>Touch Probe Cycles: Automatic Workpiece Inspection.....</b>	<b>235</b>
<b>6.1</b>	<b>Fundamentals.....</b>	<b>236</b>
	Overview.....	236
	Recording the results of measurement.....	237
	Measurement results in Q parameters.....	239
	Classification of results.....	239
	Tolerance monitoring.....	239
	Tool monitoring.....	239
	Reference system for measurement results.....	240
<b>6.2</b>	<b>Cycle 0 REF. PLANE.....</b>	<b>241</b>
	Cycle parameters.....	242
<b>6.3</b>	<b>Cycle 1 POLAR PRESET.....</b>	<b>243</b>
	Cycle parameters.....	244
<b>6.4</b>	<b>Cycle 420 MEASURE ANGLE.....</b>	<b>245</b>
	Cycle parameters.....	246
<b>6.5</b>	<b>Cycle 421 MEASURE HOLE.....</b>	<b>248</b>
	Cycle parameters.....	250
<b>6.6</b>	<b>Cycle 422 MEAS. CIRCLE OUTSIDE.....</b>	<b>253</b>
	Cycle parameters.....	255
<b>6.7</b>	<b>Cycle 423 MEAS. RECTAN. INSIDE.....</b>	<b>258</b>
	Cycle parameters.....	260
<b>6.8</b>	<b>Cycle 424 MEAS. RECTAN. OUTS.....</b>	<b>263</b>
	Cycle parameters.....	264
<b>6.9</b>	<b>Cycle 425 MEASURE INSIDE WIDTH.....</b>	<b>267</b>
	Cycle parameters.....	268
<b>6.10</b>	<b>Cycle 426 MEASURE RIDGE WIDTH.....</b>	<b>271</b>
	Cycle parameters.....	272
<b>6.11</b>	<b>Cycle 427 MEASURE COORDINATE.....</b>	<b>275</b>
	Cycle parameters.....	277
<b>6.12</b>	<b>Cycle 430 MEAS. BOLT HOLE CIRC.....</b>	<b>280</b>
	Cycle parameters.....	282
<b>6.13</b>	<b>Cycle 431 MEASURE PLANE.....</b>	<b>285</b>
	Cycle parameters.....	287

<b>6.14 Programming examples.....</b>	<b>289</b>
Example: Measuring and reworking a rectangular stud.....	289
Example: Probing a rectangular pocket and recording the results.....	291

<b>7</b>	<b>Touch Probe Cycles: Special Functions.....</b>	<b>293</b>
<b>7.1</b>	<b>Fundamentals.....</b>	<b>294</b>
	Overview.....	294
<b>7.2</b>	<b>Cycle 3 MEASURING.....</b>	<b>295</b>
	Cycle parameters.....	296
<b>7.3</b>	<b>Cycle 4 MEASURING IN 3-D.....</b>	<b>298</b>
	Cycle parameters.....	299
<b>7.4</b>	<b>Cycle 441 FAST PROBING.....</b>	<b>300</b>
	Cycle parameters.....	301
<b>7.5</b>	<b>Cycle 1493 EXTRUSION PROBING.....</b>	<b>303</b>
	Cycle parameters.....	307
<b>7.6</b>	<b>Calibrating a touch trigger probe.....</b>	<b>308</b>
<b>7.7</b>	<b>Displaying calibration values.....</b>	<b>309</b>
<b>7.8</b>	<b>Cycle 461 TS CALIBRATION OF TOOL LENGTH.....</b>	<b>310</b>
<b>7.9</b>	<b>Cycle 462 CALIBRATION OF A TS IN A RING.....</b>	<b>312</b>
<b>7.10</b>	<b>Cycle 463 TS CALIBRATION ON STUD.....</b>	<b>315</b>
<b>7.11</b>	<b>Cycle 460 CALIBRATION OF TS ON A SPHERE.....</b>	<b>318</b>

<b>8</b>	<b>Touch Probe Cycles: Automatic Tool Measurement.....</b>	<b>325</b>
<b>8.1</b>	<b>Fundamentals.....</b>	<b>326</b>
	Overview.....	326
	Differences between Cycles 31 to 33 and Cycles 481 to 483.....	327
	Measuring a tool of length 0.....	328
	Setting machine parameters.....	329
	Entries in the tool table for milling tools.....	331
<b>8.2</b>	<b>Cycle 30 or 480 CALIBRATE TT.....</b>	<b>333</b>
	Cycle parameters.....	335
<b>8.3</b>	<b>Cycle 31 or 481 CAL. TOOL LENGTH.....</b>	<b>336</b>
	Cycle parameters.....	338
<b>8.4</b>	<b>Cycle 32 or 482 CAL. TOOL RADIUS.....</b>	<b>340</b>
	Cycle parameters.....	343
<b>8.5</b>	<b>Cycle 33 or 483 MEASURE TOOL.....</b>	<b>345</b>
	Cycle parameters.....	348
<b>8.6</b>	<b>Cycle 484 CALIBRATE IR TT.....</b>	<b>350</b>
	Cycle parameters.....	353

<b>9</b>	<b>Cycles: Special Functions.....</b>	<b>355</b>
<b>9.1</b>	<b>Fundamentals.....</b>	<b>356</b>
	Overview.....	356
<b>9.2</b>	<b>Cycle 13 ORIENTATION.....</b>	<b>357</b>
	Cycle parameters.....	357

<b>10 Tables of Cycles.....</b>	<b>359</b>
<b>10.1 Table of cycles.....</b>	<b>360</b>
Touch probe cycles.....	360



# 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](mailto: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 320	771851-18
TNC 320 programming station	771855-18

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:  
1303429-xx



### User's Manual:

All control functions not related to the cycles are described in the TNC 320 User's Manual. This manual is available from HEIDENHAIN upon request.

Klartext Programming User's Manual ID: 1096950-xx

ISO Programming User's Manual ID: 1096983-xx

User's Manual for Setup, Testing and Running NC programs  
ID: 1263173-xx

## Software options

The TNC 320 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 and option 1)

<b>Additional axis</b>	Additional control loops 1 and 2
------------------------	----------------------------------

---

### Advanced Function Set 1 (option 8)

<b>Advanced functions (set 1)</b>	<p><b>Machining with rotary tables</b></p> <ul style="list-style-type: none"> <li>■ Cylindrical contours as if in two axes</li> <li>■ Feed rate in distance per minute</li> </ul> <p><b>Coordinate conversions:</b> Tilting the working plane</p> <p><b>Interpolation:</b> Circular in 3 axes with tilted working plane</p>
-----------------------------------	---

---

### HEIDENHAIN DNC (option 18)

	Communication with external PC applications over COM component
--	--

---

### CAD Import (option 42)

<b>CAD import</b>	<ul style="list-style-type: none"> <li>■ Support for DXF, STL, STEP and IGES</li> <li>■ Adoption of contours and point patterns</li> <li>■ Simple and convenient specification of presets</li> <li>■ Selecting graphical features of contour sections from conversational programs</li> </ul>
-------------------	---

---

### Extended Tool Management (option 93)

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

---

### Remote Desktop Manager (option 133)

<b>Remote operation of external computer units</b>	<ul style="list-style-type: none"> <li>■ Windows on a separate computer unit</li> <li>■ Incorporated in the control's interface</li> </ul>
--	--

---

### CAD Model Optimizer (option 152)

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

---

### Opt. contour milling (option 167)

<b>Optimized contour cycles</b>	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.

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

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 cycle functions of software 77185x-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 320. 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 cycle functions of software 77185x-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: 1322093-xx

### New cycle functions with 77185x-18

- Cycle **224 DATAMATRIX CODE PATTERN** (ISO: **G224**)  
This cycle allows the control to convert a text into a machine-readable Data Matrix code. The Data Matrix code functions as a point pattern for a previously defined machining cycle.
- Cycle **271 OCM CONTOUR DATA** (ISO: **G271**, option 167)  
This cycle allows you to define the machining information for the OCM cycles. You can define the first pocket of the contour description as an open frame. As a result, the downfeed is performed outside of the material during machining.
- Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167)  
This cycle allows the control to machine the defined contour using trochoidal milling. The control precisely adheres to the programmed path overlap, while keeping a constant tool contact angle and depth of cut.
- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**, option 167)  
This cycle allows the control to finish the contour floor with a constant path overlap. The finishing allowance for the side is left over.
- Cycle **274 OCM FINISHING SIDE** (ISO: **G274**, option 167)  
This cycle enables finishing along the contour. The control approaches the contour tangentially and machines every subcontour separately. The control does not take into account the finishing allowance defined for the floor in Cycle **271 (G271)**, but moves to full depth.
- Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167)  
This cycle allows the control to deburr contours that were last defined, roughed, or finish-machined with other OCM cycles.
- Cycle **1271 OCM RECTANGLE** (ISO: **G1271**, option 167)  
This cycle allows you to define a rectangle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1272 OCM CIRCLE** (ISO: **G1272**, option 167)  
This cycle allows you to define a circle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1273 OCM SLOT / RIDGE** (ISO: **G1273**, option 167)  
This cycle allows you to define a slot that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- 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.

- Cycle **1278 OCM POLYGON** (ISO: **G1278**, option 167)  
This cycle allows you to define a polygon that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1281 OCM RECTANGLE BOUNDARY** (ISO: **G1281**, option 167)  
This cycle allows you to define a rectangular boundary for islands or open pockets that you previously programmed with the standard OCM shapes.
- Cycle **1282 OCM CIRCLE BOUNDARY** (ISO: **G1282**, option 167)  
This cycle allows you to define a rectangular boundary for islands or open pockets that you previously programmed with the standard OCM shapes.
- The control offers an **OCM cutting data calculator** that allows you to determine the optimum cutting data for Cycle **272 OCM ROUGHING**(ISO: **G272**, option 167). Press the **OCM CUTTING DATA** soft key to open the cutting data calculator during cycle definition. You can load the results directly into the cycle parameters.

**Further information:** User's Manual for **Programming of Machining Cycles**

### New measuring cycle functions with 77185x-18

- **Cycle 1400 POSITION PROBING (ISO: G1400)**  
This cycle allows you to probe a single position. You can transfer the acquired values to the active row of the preset table.  
**Further information:** "Cycle 1400 POSITION PROBING", Page 137
- **Cycle 1401 CIRCLE PROBING (ISO: G1401)**  
This cycle allows you to determine the center point of a hole or a stud. You can transfer the acquired values to the active row of the preset table.  
**Further information:** "Cycle 1401 CIRCLE PROBING", Page 141
- **Cycle 1402 SPHERE PROBING (ISO: G1402)**  
This cycle allows you to determine the center point of a sphere. You can transfer the acquired values to the active row of the preset table.  
**Further information:** "Cycle 1402 SPHERE PROBING", Page 146
- **Cycle 1404 PROBE SLOT/RIDGE (ISO: G1404)**  
This cycle determines the center and the width of a slot or ridge. The control probes two opposing touch points. You can also define a rotation for the slot or the ridge.  
**Further information:** "Cycle 1404 PROBE SLOT/RIDGE", Page 151
- **Cycle 1412 INCLINED EDGE PROBING (ISO: G1412)**  
This cycle allows you to ascertain a workpiece misalignment by probing two points on an inclined edge.  
**Further information:** "Cycle 1412 INCLINED EDGE PROBING", Page 89
- **Cycle 1416 INTERSECTION PROBING (ISO: G1416)**  
This cycle allows you to determine the intersection of two edges. The cycle requires a total of four touch points and two positions per edge. You can use the cycle in the three object planes **XY**, **XZ** and **YZ**.  
**Further information:** "Cycle 1416 INTERSECTION PROBING", Page 97
- **Cycle 1430 PROBE POSITION OF UNDERCUT (ISO: G1430)**  
This cycle determines a single position with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus.  
**Further information:** "Cycle 1430 PROBE POSITION OF UNDERCUT", Page 155

- Cycle **1434 PROBE SLOT/RIDGE UNDERCUT** (ISO: **G1434**)  
This cycle determines the center and the width of a slot or ridge with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes two opposing touch points.  
**Further information:** "Cycle 1434 PROBE SLOT/RIDGE UNDERCUT", Page 161
- Cycle **1493 EXTRUSION PROBING** (ISO: **G1493**)  
This cycle allows you to define an extrusion. With an active extrusion, the control repeats the touch points along a direction over a defined length.  
**Further information:** "Cycle 1493 EXTRUSION PROBING", Page 303

## Modified cycle functions with 77185x-18

- Within the **CONTOUR DEF** function, you can exclude certain areas **V** (void) from machining. These areas may be contours in cast parts or machining operations from previous steps, for example.
- You can also define subcontours as **LBL** subprograms within the complex **SEL CONTOUR** contour formula.
- In Cycle **12 PGM CALL** (ISO: G39) you can use the **SYNTAX** soft key to place paths within quotation marks. To separate folders and files within paths, both the **\** and the **/** character are permitted.
- The parameter **Q357 CLEARANCE TO SIDE** has been added to Cycle **202 BORING** (ISO: **G202**). This parameter allows you to define how far the control retracts the tool at the bottom of the hole in the working plane. This parameter is only effective if parameter **Q214 DISENGAGING DIRECTN** has been defined.
- The parameter **Q357 CLEARANCE TO SIDE** has been added to Cycle **202 BORING** (ISO: **G202**). This parameter allows you to define how far the control retracts the tool at the bottom of the hole in the working plane. This parameter is only effective if parameter **Q214 DISENGAGING DIRECTN** has been defined.
- The parameter **Q373 FEED AFTER REMOVAL** has been added to Cycle **205 UNIVERSAL PECKING** (ISO: **G205**). This parameter is used to define the feed rate for returning to the advanced stop distance after chip removal.
- Cycles **205 UNIVERSAL PECKING** (ISO: **G205**) and **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**) check the parameter **Q379 STARTING POINT**. If the value of the starting point is equal to or greater than the value of the parameter **Q201 DEPTH**, then the control issues an error message.
- The parameters **Q429 COOLANT ON** and **Q430 COOLANT OFF** in Cycle **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**) have been enhanced. You can define a path for a user macro.
- The parameter **Q370 TOOL PATH OVERLAP** has been added to Cycle **208 BORE MILLING** (ISO: **G208**). Use this parameter to define the lateral infeed.
- Cycle **240 CENTERING** (ISO: **G240**) has been extended in order to consider pre-drilled diameters.  
The following parameters have been added:
  - **Q342 ROUGHING DIAMETER**
  - **Q253 F PRE-POSITIONING**: If parameter **Q342** is defined, feed rate for approaching the deepened starting point
- 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.

- Cycle **225 ENGRAVING** (ISO: **G225**) has been expanded:
  - The parameter **Q202 MAX. PLUNGING DEPTH** allows you to define the maximum plunging depth.
  - The parameter **Q367 TEXT POSITION** now includes the input options **7**, **8** and **9**. With these values, you can set the reference of the engraving text onto the horizontal centerline.
  - The approach behavior was changed. If the tool is below the **2ND SET-UP CLEARANCE**, the control first positions the tool to the 2nd set-up clearance **Q204** and then to the starting position in the working plane.
- 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 Cycle **225 ENGRAVING** (ISO: **G225**), you can program system variables that contain the following information for the current main program and the NC program called:
  - Entire file path
  - Directory path
  - File name
  - File type
- Cycle **225 ENGRAVING** (ISO: **G225**) allows you to use system variables to engrave the number of the current week.
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), you program a limit that is perpendicular to the milling direction **Q350**, then the control adds the tool radius to the length of the surface in the unlimited direction. As a result, the control completely machines the defined surface without leaving behind any residual material, as would be caused by the tool radius. If the parameter **Q220** (corner radius) is defined, then the control adds both the tool radius and this value to the length of the surface.
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), the parameter **Q389** has been defined with the value 2 or 3 and a lateral limit is defined in addition, then the control approaches the contour or departs from it on an arc with **Q207 FEED RATE MILLING**.
- Cycles **208 BORE MILLING** (ISO: **G208**), **253 SLOT MILLING** (ISO: **G208**) and **254 CIRCULAR SLOT** (ISO: **G254**) monitor a cutting width defined in the column **RCUTS** of the tool table. If the center of a tool that is not a center-cut tool would contact the workpiece surface, the control issues an error message.
- Cycles **251 RECTANGULAR POCKET** (ISO: **G251**), **252 CIRCULAR POCKET** (ISO: **G252**) and **272 OCM ROUGHING** (ISO: **G272**, option 167) take into consideration a cutting width defined in the column **RCUTS** when calculating the plunging path.

- If the defined usable length in column **LU** of the tool table is less than the depth, the control displays an error message.

The following cycles monitor the usable length LU:

- All cycles for drilling and boring
- All cycles for tapping
- All cycles for the machining of pockets and studs
- Cycle 22 **ROUGH-OUT** (ISO: **G122**)
- Cycle 23 **FLOOR FINISHING** (ISO: **G123**)
- Cycle 24 **SIDE FINISHING** (ISO: **G124**)
- Cycle 233 **FACE MILLING** (ISO: **G233**)
- Cycle 272 **OCM ROUGHING** (ISO: **G272**, option 167)
- Cycle 273 **OCM FINISHING FLOOR** (ISO: **G273**, option 167)
- Cycle 274 **OCM FINISHING SIDE** (ISO: **G274**, option 167)
- Certain cycles permit you to enter tolerances. In the following cycles, you can define dimensions, tolerances according to EN ISO 286-2 or general tolerances according to ISO 2768-1:
  - Cycle **208 BORE MILLING** (ISO: **G208**)
  - **127x** (Option 167) OCM standard figure cycles
- The following cycles consider the miscellaneous functions **M109** and **M110**:
  - Cycle **22 ROUGH-OUT** (ISO: G122)
  - Cycle **23 FLOOR FINISHING** (ISO: G123)
  - Cycle **24 SIDE FINISHING** (ISO: G124)
  - Cycle **25 CONTOUR TRAIN** (ISO: G125)
  - Cycle **275 TROCHOIDAL SLOT** (ISO: G275)
  - Cycle **276 THREE-D CONT. TRAIN** (ISO: G276)
  - Cycle **274 OCM FINISHING SIDE** (ISO: G274, option 167)
  - Cycle **277 OCM CHAMFERING** (ISO: G277, option 167)

#### **Further information:** User's Manual for **Programming of Machining Cycles**

- Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**) determines the radius, length, center offset and spindle angle of an L stylus.  
**Further information:** "Cycle 460 CALIBRATION OF TS ON A SPHERE", Page 318
- Cycles **14xx** support probing with an L-shaped stylus.  
**Further information:** "Working with an L-shaped stylus", Page 41
- The unit of measure of the main program can be seen in the header of the log file of the Probing Cycles **14xx** and **42x**  
**Further information:** "Characteristics common to touch probe cycles 14xx for measuring rotations", Page 55  
**Further information:** "Recording the results of measurement", Page 237



- In cycles with the number **14xx**, you can now pre-position with a handwheel in semi-automatic mode. After probing, you can move to clearance height manually.

**Further information:** "Semi-automatic mode", Page 57

- The Cycles **1420 PROBING IN PLANE** (ISO: **G1420**), **1410 PROBING ON EDGE** (ISO: **G1410**), **1411 PROBING TWO CIRCLES** (ISO: **G1411**) have been enhanced:
  - In these cycles, you can define tolerances according to EN ISO 286-2 or general tolerances according to ISO 2768-1.
  - If you defined the value 2 in parameter **Q1125 CLEAR. HEIGHT MODE**, the control positions the touch probe at rapid traverse **FMAX** from the touch probe table to the set-up clearance.

**Further information:** "Evaluation of tolerances", Page 62

- By default, Cycles **1410 PROBING ON EDGE** (ISO: **G1410**) and **1411 PROBING TWO CIRCLES** (ISO: **G1411**) calculate the basic rotation in the input coordinate system (I-CS). If the axis angles and tilt angles don't coincide, the cycles calculate the basic rotation in the workpiece coordinate system (W-CS).

**Further information:** "Cycle 1410 PROBING ON EDGE", Page 73

**Further information:** "Cycle 1411 PROBING TWO CIRCLES", Page 81

- 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 300

- 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:
  - 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 300

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**) allow you to calibrate a tool touch probe with cuboid probe contacts.  
**Further information:** "Cycle 30 or 480 CALIBRATE TT", Page 333  
**Further information:** "Cycle 484 CALIBRATE IR TT", Page 350
- The parameter **Q523 TT-POSITION** has been added to Cycle **484 CALIBRATE IR TT** (ISO: **G484**). This parameter allows you to define the position of the tool touch probe and, if desired, to transfer the position to the machine parameter **centerPos** after calibration.  
**Further information:** "Cycle 484 CALIBRATE IR TT", Page 350
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**) first measures the tool length and then the tool radius.  
**Further information:** "Cycle 33 or 483 MEASURE TOOL", Page 345
- 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 328
- 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 329

# 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	<b>Further information:</b> User's Manual for Programming of Machining Cycles
	Cycles for tapping, thread cutting and thread milling	<b>Further information:</b> User's Manual for Programming of Machining Cycles
	Cycles for milling pockets, studs, slots, and face milling	<b>Further information:</b> User's Manual for Programming of Machining Cycles
	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	<b>Further information:</b> 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	<b>Further information:</b> User's Manual for Programming of Machining Cycles
	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	<b>Further information:</b> User's Manual for Programming of Machining Cycles
	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance,	<b>Further information:</b> User's Manual for Programming of Machining Cycles
	<ul style="list-style-type: none"> <li>▶ If required, switch to machine-specific machining cycles</li> </ul> <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	54
	Cycles for automatic workpiece presetting	134
	Cycles for automatic workpiece inspection	236
	Special cycles	294
	Touch probe calibration	308
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	326
	▶ 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.

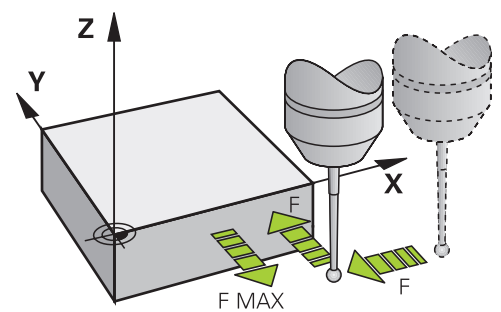


HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

#### 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 44

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 308



### 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

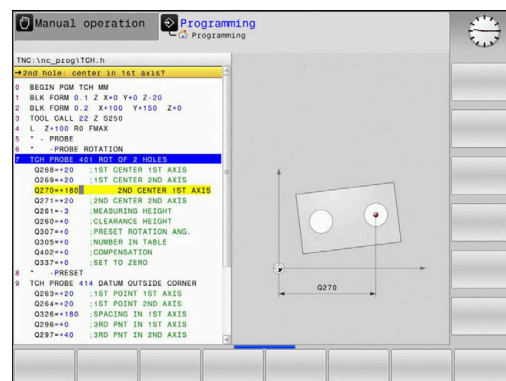
## Touch probe cycles for automatic operation

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

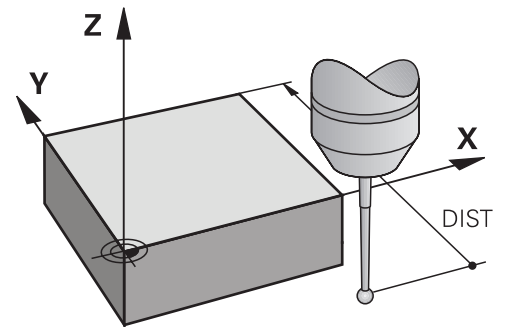
## 3.2 Before you start working with touch probe cycles

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

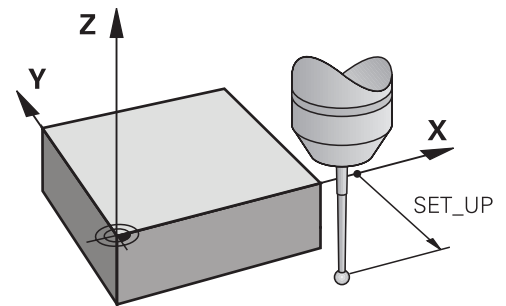
### Maximum traverse to touch point: **DIST** in touch probe table

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**.

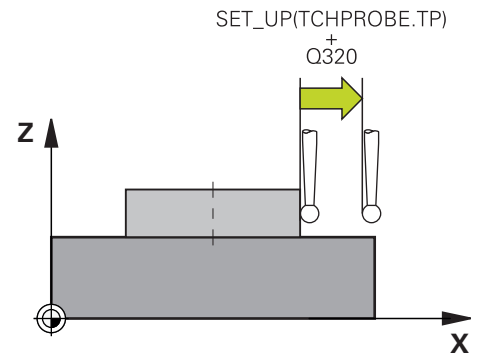
### Pre-positioning

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

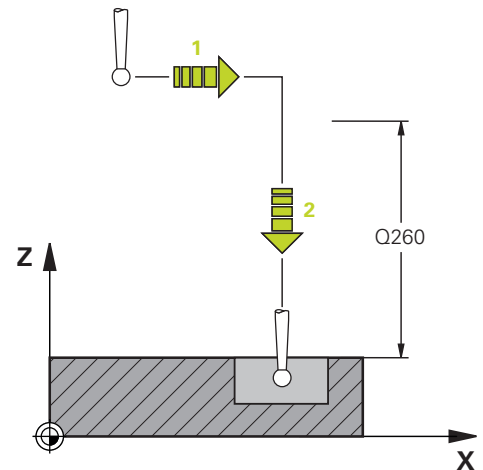
Touch-probe cycles with numbers from **400** through **499** or **1400** through **1499** pre-position the touch probe according to the following positioning logic:

#### 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 47

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



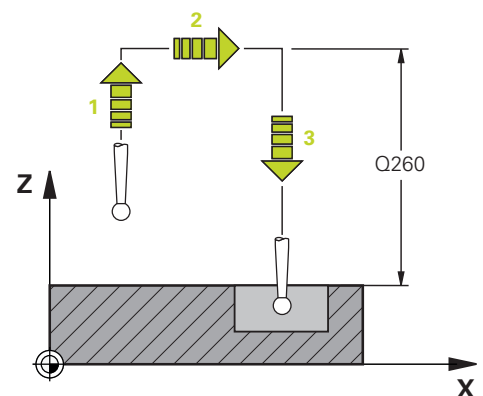
#### Current position < Q260 CLEARANCE HEIGHT

- 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 47

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

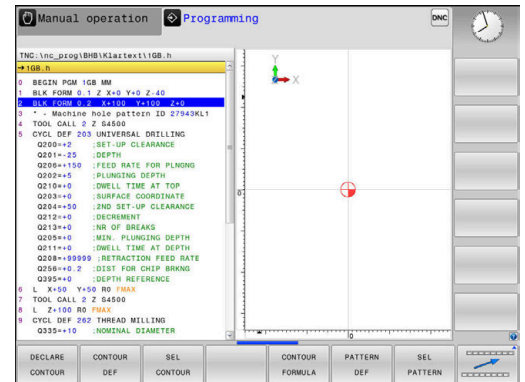


### 3.3 Program defaults for cycles

#### 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:

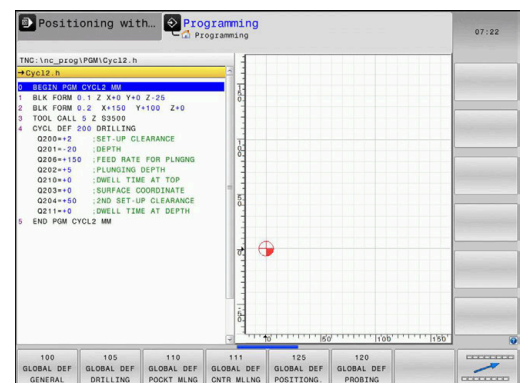


Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	<b>GLOBAL DEF GENERAL</b> Definition of generally valid cycle parameters	50
105 GLOBAL DEF DRILLING	<b>GLOBAL DEF DRILLING</b> Definition of specific drilling cycle parameters	<b>Further information:</b> User's Manual for Programming of Machining Cycles
110 GLOBAL DEF POCKET MILLNG	<b>GLOBAL DEF POCKET MILLING</b> Definition of specific pocket-milling cycle parameters	<b>Further information:</b> User's Manual for Programming of Machining Cycles
111 GLOBAL DEF CNTR MILLNG	<b>GLOBAL DEF CONTOUR MILLING</b> Definition of specific contour milling cycle parameters	<b>Further information:</b> User's Manual for Programming of Machining Cycles
125 GLOBAL DEF POSITIONING	<b>GLOBAL DEF POSITIONING</b> Definition of the positioning behavior with <b>CYCL CALL PAT</b>	<b>Further information:</b> User's Manual for Programming of Machining Cycles
120 GLOBAL DEF PROBING	<b>GLOBAL DEF PROBING</b> Definition of specific touch probe cycle parameters	51

#### 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








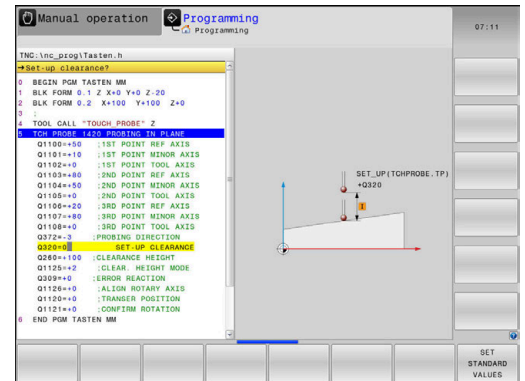


### Using GLOBAL DEF information

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

Help graphic	Parameter
	<p><b>Q200 Set-up clearance?</b> Distance between tool tip and workpiece surface. This value has an incremental effect. Input: <b>0...99999.9999</b></p>
	<p><b>Q204 2nd set-up clearance?</b> 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: <b>0...99999.9999</b></p>
	<p><b>Q253 Feed rate for pre-positioning?</b> Feed rate at which the control moves the tool within a cycle. Input: <b>0...99999.999</b> or <b>FMAX, FAUTO</b></p>
	<p><b>Q208 Feed rate for retraction?</b> Feed rate at which the control retracts the tool. Input: <b>0...99999.999</b> or <b>FMAX, FAUTO</b></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, 1271, 1272, 1273, 1274, 1278**

Help graphic	Parameter
	<p><b>Q320 Set-up clearance?</b></p> <p>Additional distance between touch point and ball tip. <b>Q320</b> is active in addition to the <b>SET_UP</b> column in the touch probe table. This value has an incremental effect.</p> <p>Input: <b>0...99999.9999</b> or <b>PREDEF</b></p>
	<p><b>Q260 Clearance height?</b></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: <b>-99999.9999...+99999.9999</b> or <b>PREDEF</b></p>
	<p><b>Q301 Move to clearance height (0/1)?</b></p> <p>Define how the touch probe will move between the measuring points:</p> <p><b>0:</b> Move to measuring height between measuring points  <b>1:</b> Move to clearance height between measuring points</p> <p>Input: <b>0, 1</b></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.

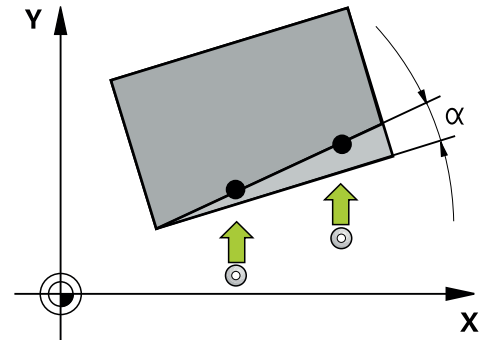
Soft key	Cycle	Page
	Cycle 1420 PROBING IN PLANE <ul style="list-style-type: none"> <li>Automatic measurement using three points</li> <li>Compensation via basic rotation or rotary table rotation</li> </ul>	66
	Cycle 1410 PROBING ON EDGE <ul style="list-style-type: none"> <li>Automatic measurement using two points</li> <li>Compensation via basic rotation or rotary table rotation</li> </ul>	73
	Cycle 1411 PROBING TWO CIRCLES <ul style="list-style-type: none"> <li>Automatic measurement using two holes or studs</li> <li>Compensation via basic rotation or rotary table rotation</li> </ul>	81
	Cycle 1412 INCLINED EDGE PROBING <ul style="list-style-type: none"> <li>Automatic measurement using two points on an inclined edge</li> <li>Compensation via basic rotation or rotary table rotation</li> </ul>	89
	Cycle 1416 INTERSECTION PROBING <ul style="list-style-type: none"> <li>Automatically determines the intersection with four touch points on two straight lines</li> <li>Compensation via basic rotation or rotary table rotation</li> </ul>	97
	Cycle 400 BASIC ROTATION <ul style="list-style-type: none"> <li>Automatic measurement using two points</li> <li>Compensation via basic rotation</li> </ul>	107
	Cycle 401 ROT OF 2 HOLES <ul style="list-style-type: none"> <li>Automatic measurement using two holes</li> <li>Compensation via basic rotation</li> </ul>	111
	Cycle 402 ROT OF 2 STUDS <ul style="list-style-type: none"> <li>Automatic measurement using two studs</li> <li>Compensation via basic rotation</li> </ul>	116
	Cycle 403 ROT IN ROTARY AXIS <ul style="list-style-type: none"> <li>Automatic measurement using two points</li> <li>Compensation via rotary table rotation</li> </ul>	121
	Cycle 405 ROT IN C AXIS <ul style="list-style-type: none"> <li>Automatic alignment of an angular offset between a hole center and the positive Y axis</li> <li>Compensation via rotary table rotation</li> </ul>	126
	Cycle 404 SET BASIC ROTATION <ul style="list-style-type: none"> <li>Setting any basic rotation</li> </ul>	130

## 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: <b>Input Coordinate System</b>
W-CS	W-CS: <b>Workpiece Coordinate System</b>
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.



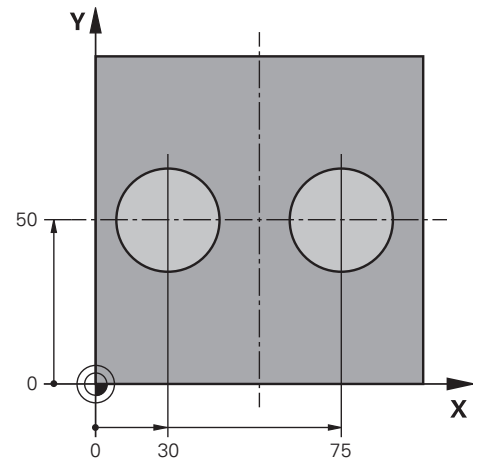
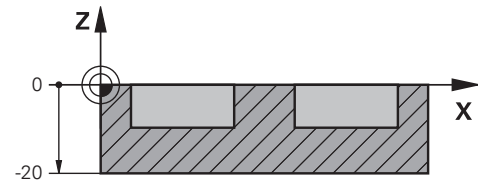
Programming and operating notes:

- 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**

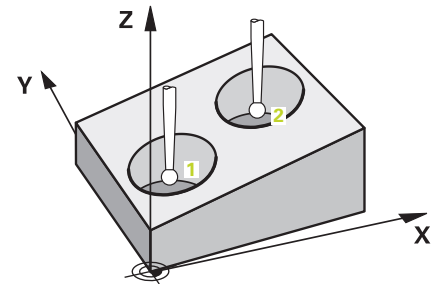
**Important:** Specify the **nominal positions** from the drawing!

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



## Hole

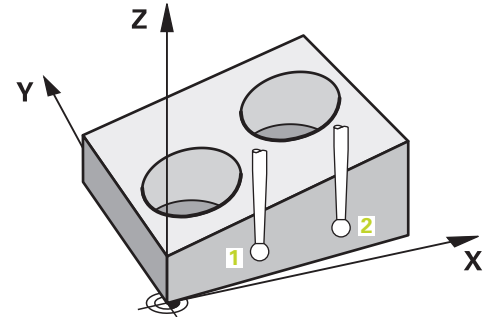
In this example, you will align two holes. Probing is done in the X axis (main axis) and in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure in this direction.



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

## Edge

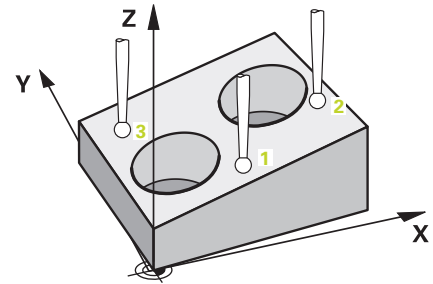
In this example, you will align an edge. Probing is done in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for this axis! Nominal positions for the X axis (main axis) and for the Z axis (tool axis) are not required because you will not measure in these directions.



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.



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

## 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.

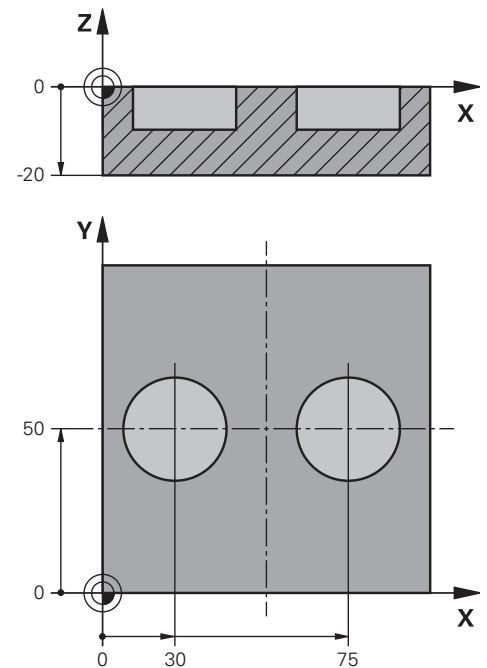
#### If Q309 = 1 or 2:

- 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



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



## 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.

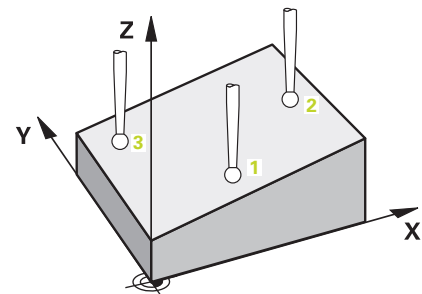
**Further information:** "Cycle 1493 EXTRUSION PROBING", Page 303

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 57
- 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 62
- 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 65

### 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 47
- 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"> <li>■ -1 = Not defined</li> <li>■ 0 = Good</li> <li>■ 1 = Rework</li> <li>■ 2 = Scrap</li> <li>■ 3 = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the first touch point</p>
Q971	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the second touch point</p>
Q972	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</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.



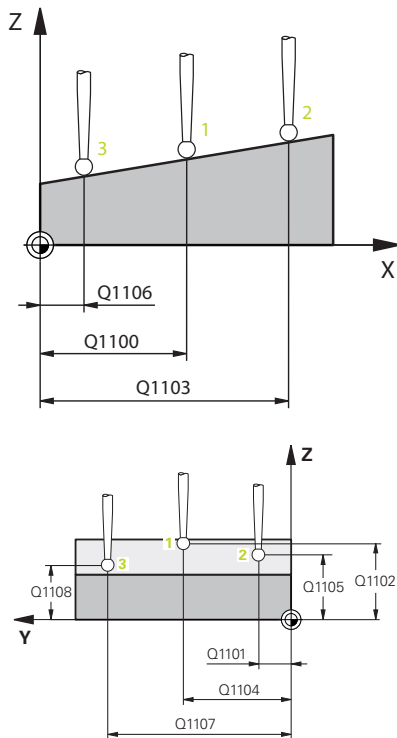
- HEIDENHAIN recommends avoiding the use of axis angles in this cycle!

#### 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 57
- **-, +**: Evaluation of the tolerance, see Page 62
- **@**: Transfer of an actual position, see Page 65

#### 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

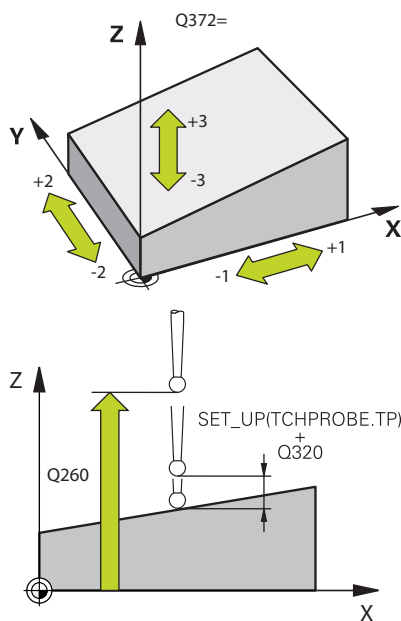
Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

#### 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 303

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 57

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

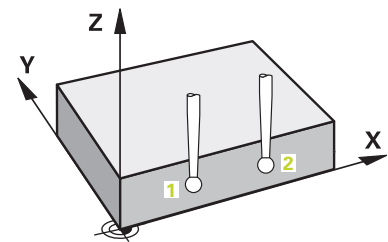
**Further information:** "Evaluation of tolerances", Page 62

- 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 65

### 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 47
- 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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the first touch point</p>
Q971	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</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.

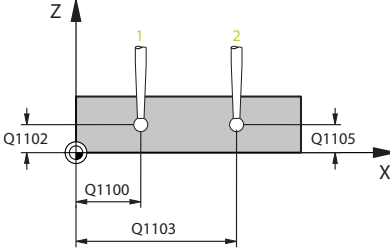
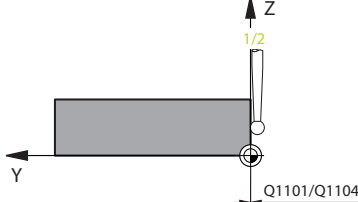
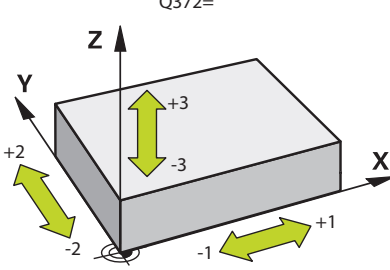
#### Note about rotary axes:

- 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**.

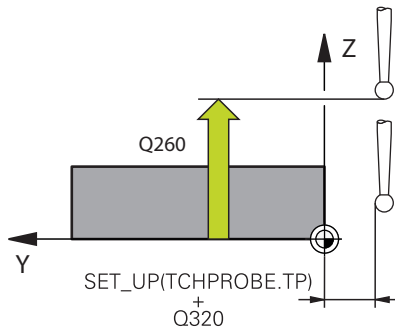
**Aligning the rotary table axes:**

- 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
	<p><b>Q1100 1st noml. position of ref. axis?</b>                      Absolute nominal position of the first touch point in the main axis of the working plane                      Input: <b>-99999.9999...+99999.9999</b> or <b>?, -, +</b> or <b>@</b></p> <ul style="list-style-type: none"> <li>■ <b>?</b>: Semi-automatic mode, see Page 57</li> <li>■ <b>-, +</b>: Evaluation of the tolerance, see Page 62</li> <li>■ <b>@</b>: Transfer of an actual position, see Page 65</li> </ul>
	<p><b>Q1101 1st noml. position of minor axis?</b>                      Absolute nominal position of the first touch point in the secondary axis of the working plane                      Input: <b>-99999.9999...+9999.9999</b> or optional input (see <b>Q1100</b>)</p>
	<p><b>Q1102 1st nominal position tool axis?</b>                      Absolute nominal position of the first touch point in the tool axis                      Input: <b>-99999.9999...+9999.9999</b> or optional input (see <b>Q1100</b>)</p>
	<p><b>Q1103 2nd noml. position of ref axis?</b>                      Absolute nominal position of the second touch point in the main axis of the working plane                      Input: <b>-99999.9999...+9999.9999</b> or optional input (see <b>Q1100</b>)</p>
	<p><b>Q1104 2nd noml. position of minor axis?</b>                      Absolute nominal position of the second touch point in the secondary axis of the working plane                      Input: <b>-99999.9999...+9999.9999</b> or optional input (see <b>Q1100</b>)</p>
	<p><b>Q1105 2nd nominal pos. of tool axis?</b>                      Absolute nominal position of the second touch point in the tool axis of the working plane                      Input: <b>-99999.9999...+9999.9999</b> or optional input (see <b>Q1100</b>)</p> <p><b>Q372 Probe direction (-3 to +3)?</b>                      Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction.                      Input: <b>-3, -2, -1, +1, +2, +3</b></p>

## 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 303

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 57

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

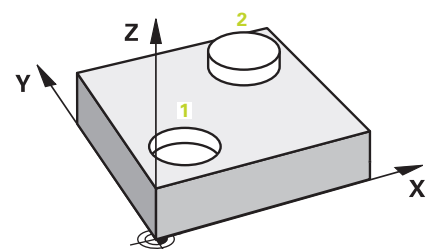
**Further information:** "Evaluation of tolerances", Page 62

- 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 65

### 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 47
- 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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the first circle center</p>
Q971	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the second circle center</p>
Q973	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from Diameter 1</p>
Q974	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</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:**

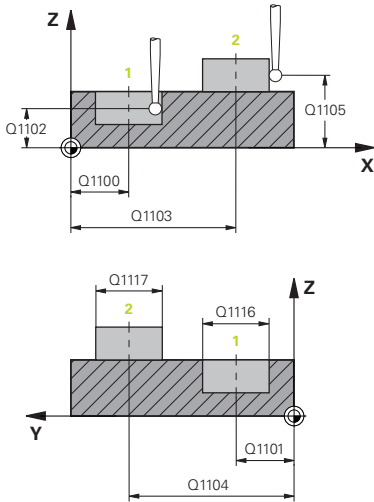
- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining plane is consistent. The control calculates the basic rotation in the input coordinate system **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 57
- **"...-...+..."**: Evaluation of the tolerance, see Page 62
- **"...@..."**: Transfer of an actual position, see Page 65

#### 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 62

#### 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 62

**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?**

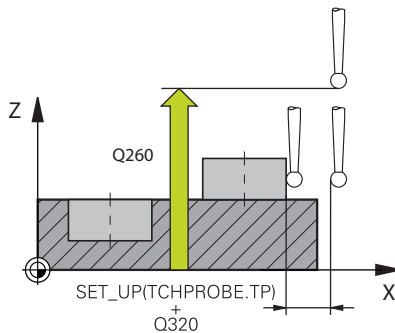
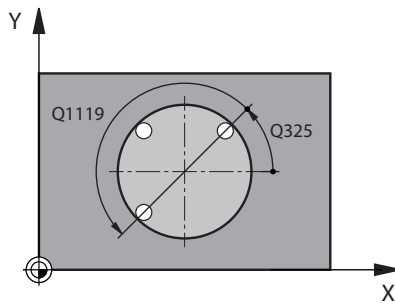
Additional distance between touch point and ball tip. **Q320** is added to **SET\_UP** (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect.

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 303

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.

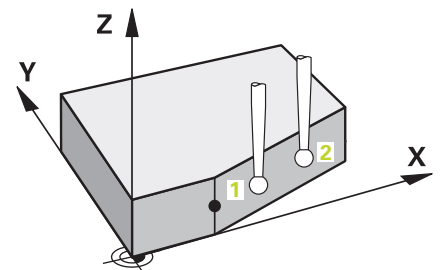
**Further information:** "Semi-automatic mode", Page 57

- 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 65

### 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 47
- 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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the first touch point</p>
Q971	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</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:

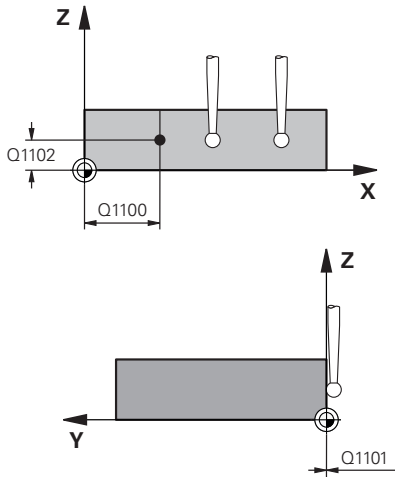
- When you determine the basic rotation in a tilted working plane, keep the following in mind:
  - If the current coordinates of the rotary axes and the defined tilt angles (3D ROT menu) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
  - 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 57
- **-, +**: Evaluation of the tolerance, see Page 62
- **@**: Transfer of an actual position, see Page 65

#### 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?

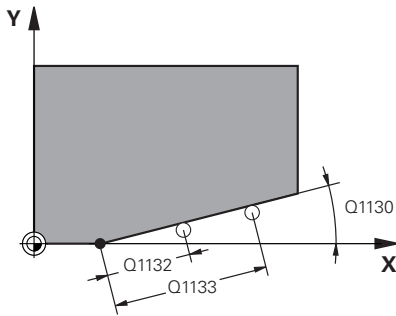
Tolerance band monitored by the cycle. The tolerance defines the deviation permitted for the surface normals along the inclined edge. The control determines this deviation using the nominal coordinate and the actual coordinate of the workpiece.

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  $-90^\circ$  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:

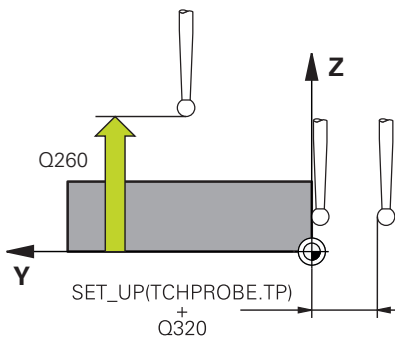
**-1**: Do not move to the clearance height.

**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 303

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.

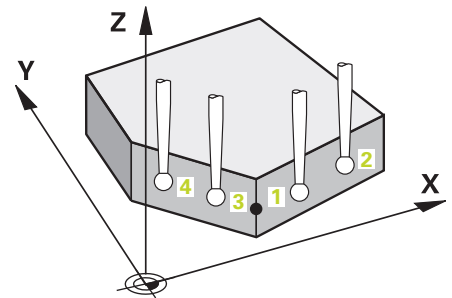
**Further information:** "Semi-automatic mode", Page 57

- 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 65

### 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 47
- 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"> <li>■ -1 = Not defined</li> <li>■ 0 = Good</li> <li>■ 1 = Rework</li> <li>■ 2 = Scrap</li> <li>■ 3 = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation from the 1st touch point</p>
Q971	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation from the 2nd touch point</p>
Q972	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</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:

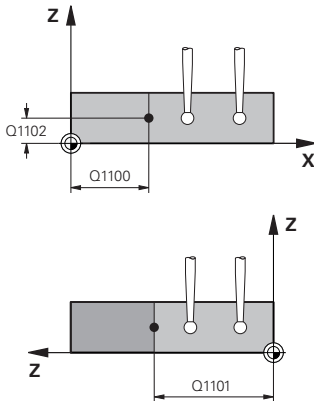
- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining plane is consistent. The control calculates the basic rotation in the input coordinate system **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 57
- @ : Transfer of an actual position, see Page 65

#### 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?

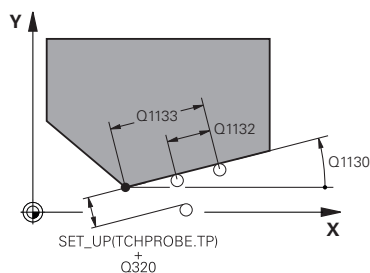
Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normal along the first edge. The control determines the deviation using the nominal coordinates and the actual coordinates of the part.

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  $-90^\circ$  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  $-90^\circ$  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?**

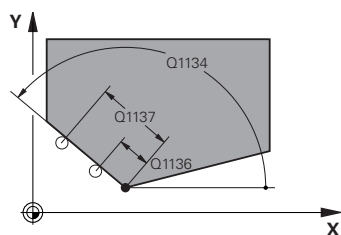
Distance between the intersection and the first touch point on the second edge. This value has an incremental effect.

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

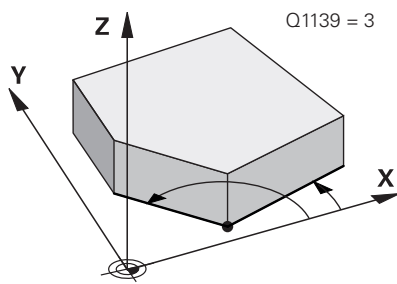
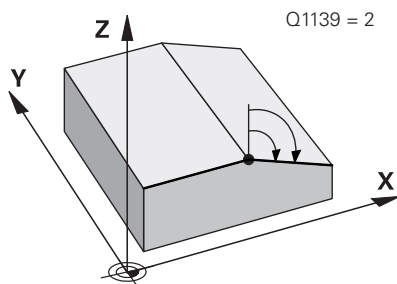
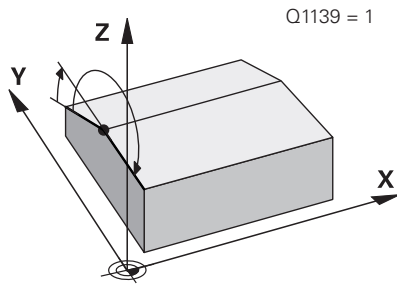
**Q1137 Second distance on 2nd line?**

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**

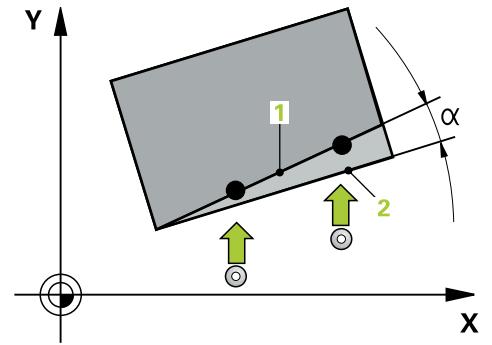
11 TCH PROBE 1416 INTERSECTION PROBING ~	
Q1100=+50	;1ST POINT REF AXIS ~
Q1101=+10	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS400="0"	;TOLERANCE ~
Q1130=+45	;NOMINAL ANGLE, 1ST LINE ~
Q1131=+1	;PROBE DIRECTION, 1ST LINE ~
Q1132=+10	;FIRST DISTANCE, 1ST LINE ~
Q1133=+25	;SECOND DISTANCE, 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  $\alpha$  (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  $0^\circ$  direction **2**.

**i** These cycles do not work with 3D ROT! In such a case, use cycles **14xx**. **Further information:** "Touch probe cycles 14xx: fundamentals", Page 55



## 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:

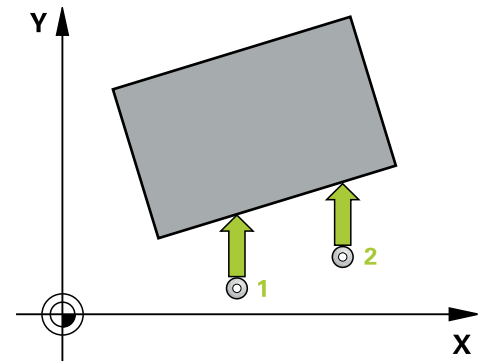
- **1410 PROBING ON EDGE**
- **1412 INCLINED EDGE PROBING**

### Related topics

- Cycle **1410 PROBING ON EDGE**  
**Further information:** "Cycle 1410 PROBING ON EDGE", Page 73
- Cycle **1412 INCLINED EDGE PROBING**  
**Further information:** "Cycle 1412 INCLINED EDGE PROBING", Page 89

### 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 47
- 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.

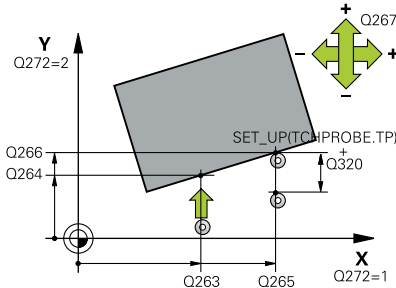
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

#### 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?

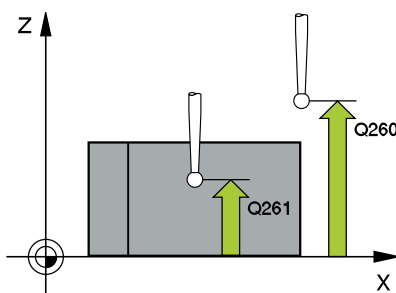
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET\_UP** column in the touch probe table. This value has an incremental effect.

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**.

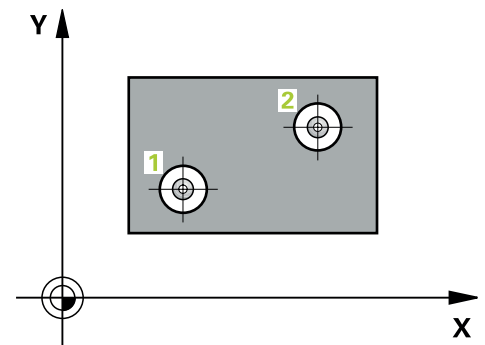
### Related topics

- Cycle **1411 PROBING TWO CIRCLES**

**Further information:** "Cycle 1411 PROBING TWO CIRCLES", Page 81

### 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 47
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.



## Notes

### NOTICE

#### Danger of collision!

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

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

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

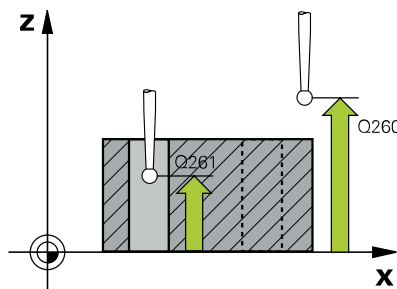
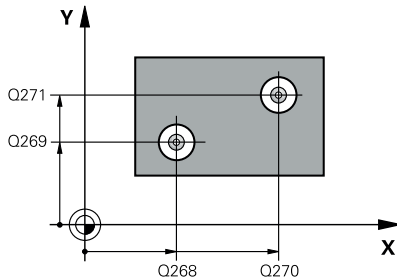
#### Note on programming

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



## Cycle parameters

### Help graphic



### Parameter

#### Q268 1st hole: center in 1st axis?

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

Input: **-99999.9999...+9999.9999**

#### Q269 1st hole: center in 2nd axis?

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

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

#### Q270 2nd hole: center in 1st axis?

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

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

#### Q271 2nd hole: center in 2nd axis?

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

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

#### Q261 Measuring height in probe axis?

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

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

#### Q260 Clearance height?

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

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

#### Q307 Preset value for rotation angle

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

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

## Help graphic

## Parameter

**Q305 Number in table?**

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

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

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

**Q305 depends on the following parameters:**

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

Input: **0...99999**

**Q402 Basic rotation/alignment (0/1)**

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

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

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

Input: **0, 1**

**Q337 Set to zero after alignment?**

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

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

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

Input: **0, 1**

**Example**

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

## 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.

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

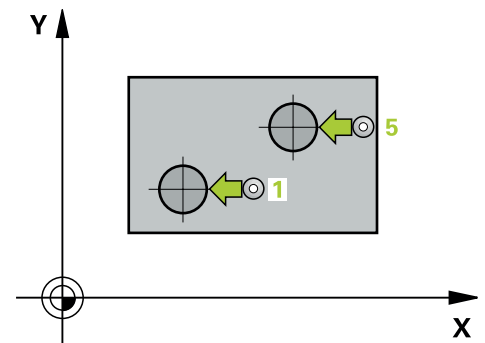
### Related topics

- Cycle **1411 PROBING TWO CIRCLES**

**Further information:** "Cycle 1411 PROBING TWO CIRCLES", Page 81

### 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 47
- 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.

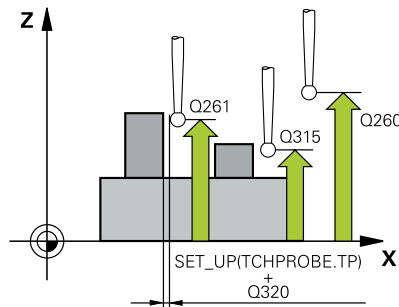
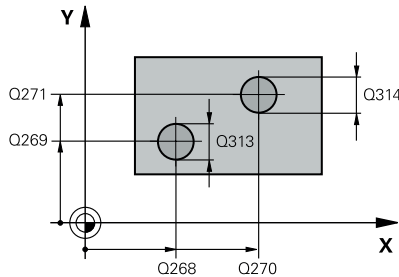
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
  -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

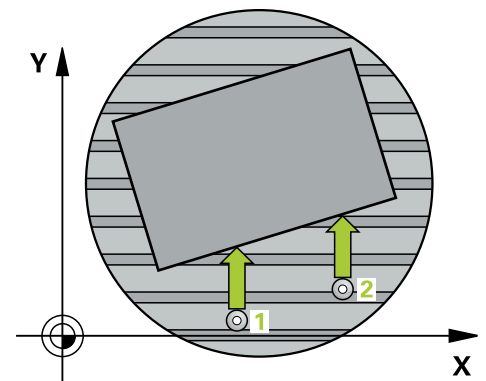
- **1410 PROBING ON EDGE**
- **1412 INCLINED EDGE PROBING**

### Related topics

- Cycle **1410 PROBING ON EDGE**  
**Further information:** "Cycle 1410 PROBING ON EDGE", Page 73
- Cycle **1412 INCLINED EDGE PROBING**  
**Further information:** "Cycle 1412 INCLINED EDGE PROBING", Page 89

### 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 47
- 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  $-90^\circ$  to  $+90^\circ$ . There is a risk of collision!

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

**NOTICE****Danger of collision!**

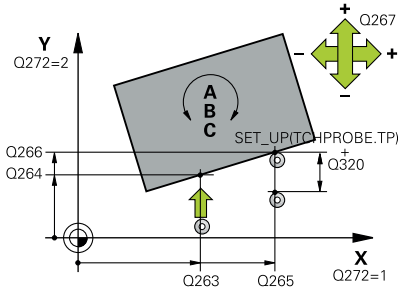
When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

- ▶ 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?

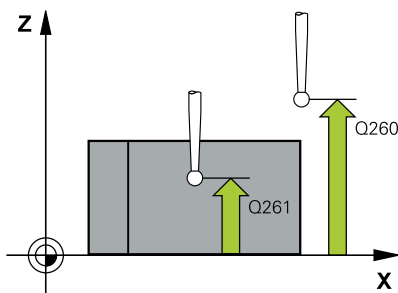
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET\_UP** column in the touch probe table. This value has an incremental effect.

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

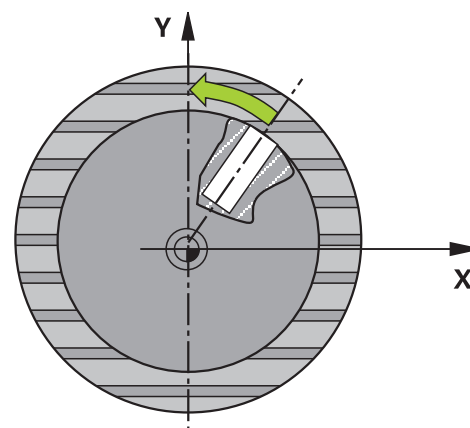
#### G405

### 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**.

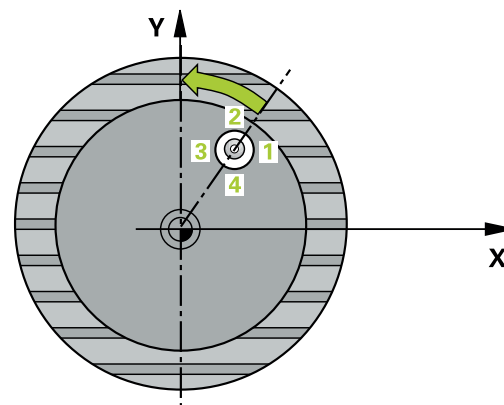
### Related topics

- Cycle **1411 PROBING TWO CIRCLES**

**Further information:** "Cycle 1411 PROBING TWO CIRCLES", Page 81

### 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 47
- 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.

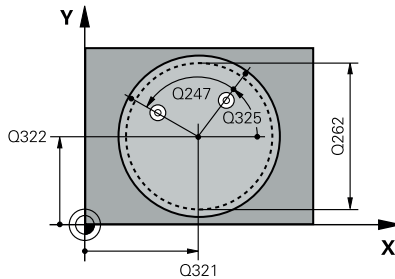
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

#### 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?

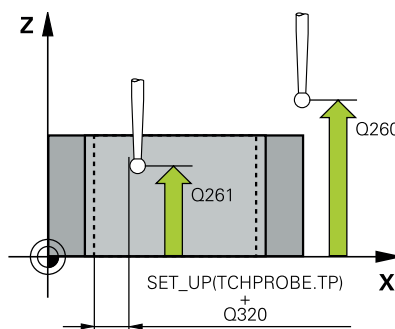
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET\_UP** column in the touch probe table. This value has an incremental effect.

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

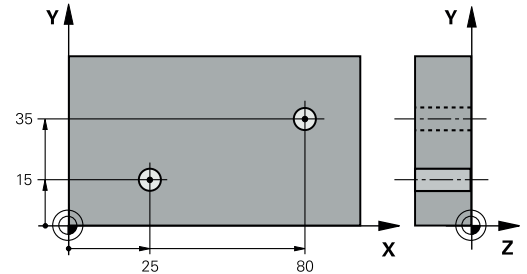
Help graphic	Parameter
	<p><b>Q307 Preset value for rotation angle</b> Angle value at which the basic rotation will be set. Input: <b>-360.000...+360.000</b></p>
	<p><b>Q305 Preset number in table?:</b> Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter <b>Q305 = 0</b> or <b>Q305 = -1</b>, the control additionally saves the calculated basic rotation in the basic rotation menu (<b>Probing rot</b>) of <b>Manual Operation</b> 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 &gt; 1: Save the basic rotation to the specified preset. The preset is not activated. Input: <b>-1...99999</b></p>

### 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



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



# 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"> <li>■ Measuring a single position</li> <li>■ Setting as preset, if applicable</li> </ul>	137
	Cycle 1401 CIRCLE PROBING <ul style="list-style-type: none"> <li>■ Measuring points on the inside or outside of a circle</li> <li>■ Setting the circle center as preset, if applicable</li> </ul>	141
	Cycle 1402 SPHERE PROBING <ul style="list-style-type: none"> <li>■ Measuring points on a sphere</li> <li>■ Setting the sphere center as preset, if applicable</li> </ul>	146
	Cycle 1404 PROBE SLOT/RIDGE <ul style="list-style-type: none"> <li>■ Measuring the center of the slot width or ridge width</li> <li>■ Setting the center as a preset if needed</li> </ul>	151
	Cycle 1430 PROBE POSITION OF UNDERCUT <ul style="list-style-type: none"> <li>■ Measuring the undercut</li> <li>■ Measuring the individual position with an L-shaped stylus</li> <li>■ Definition of preset, if necessary</li> </ul>	155
	Cycle 1434 PROBE SLOT/RIDGE UNDERCUT <ul style="list-style-type: none"> <li>■ Measuring the undercut</li> <li>■ Measuring the center of the slot width or ridge width with an L-shaped stylus</li> <li>■ Setting the center as a preset if needed</li> </ul>	161
	Cycle 410 PRESET INSIDE RECTAN <ul style="list-style-type: none"> <li>■ Measuring the inside length and width of a rectangle</li> <li>■ Setting the rectangle center as preset</li> </ul>	169
	Cycle 411 PRESET OUTS. RECTAN <ul style="list-style-type: none"> <li>■ Measuring the outside length and width of a rectangle</li> <li>■ Setting the rectangle center as preset</li> </ul>	174
	Cycle 412 PRESET INSIDE CIRCLE <ul style="list-style-type: none"> <li>■ Measuring any four points on the inside of a circle</li> <li>■ Setting the circle center as preset</li> </ul>	180
	Cycle 413 PRESET OUTS. CIRCLE <ul style="list-style-type: none"> <li>■ Measuring any four points on the outside of a circle</li> <li>■ Setting the circle center as preset</li> </ul>	186

Soft key	Cycle	Page
	Cycle 414 PRESET OUTS. CORNER <ul style="list-style-type: none"> <li>Measuring two straight lines on the outside</li> <li>Setting the intersection of the lines as preset</li> </ul>	192
	Cycle 415 PRESET INSIDE CORNER <ul style="list-style-type: none"> <li>Measuring two straight lines on the inside</li> <li>Setting the intersection of the lines as preset</li> </ul>	198
	Cycle 416 PRESET CIRCLE CENTER <ul style="list-style-type: none"> <li>Measuring any three holes of a bolt hole circle</li> <li>Setting the center of the bolt hole circle as preset</li> </ul>	204
	Cycle 417 PRESET IN TS AXIS <ul style="list-style-type: none"> <li>Measuring any position in the tool axis</li> <li>Setting any position as preset</li> </ul>	210
	Cycle 418 PRESET FROM 4 HOLES <ul style="list-style-type: none"> <li>Measuring two holes on each line crosswise</li> <li>Setting the intersection of the connecting lines as preset</li> </ul>	214
	Cycle 419 PRESET IN ONE AXIS <ul style="list-style-type: none"> <li>Measuring any position in a selectable axis</li> <li>Setting any position in a selectable axis as preset</li> </ul>	219
	Cycle 408 SLOT CENTER PRESET <ul style="list-style-type: none"> <li>Measuring the inside width of a slot</li> <li>Setting the slot center as preset</li> </ul>	223
	Cycle 409 RIDGE CENTER PRESET <ul style="list-style-type: none"> <li>Measuring the outside width of a ridge</li> <li>Setting the ridge center as preset</li> </ul>	228

## 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

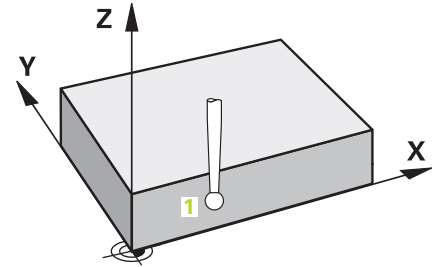
Touch probe cycle **1400** measures any position in a selectable axis. You can apply the result to the active row of the preset table.

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 303

**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 47
- 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 136



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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the first touch point</p>

Notes

**NOTICE**

**Danger of collision!**

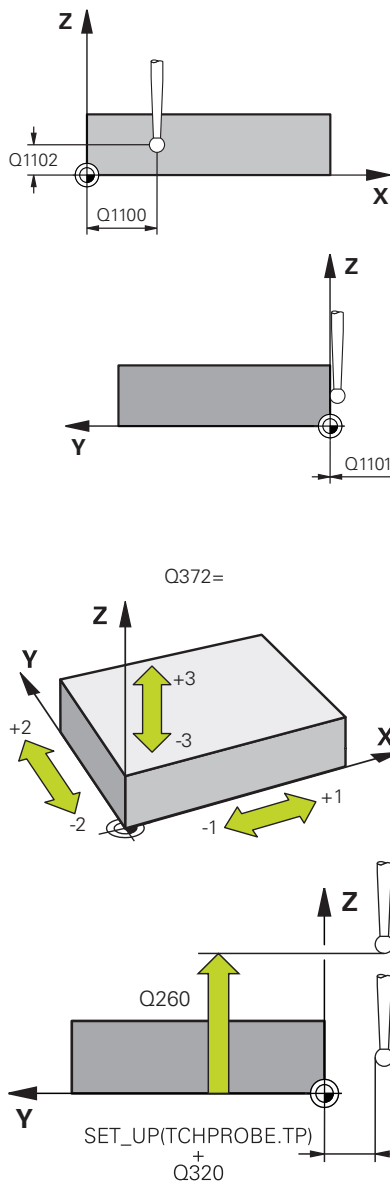
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ 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 57
- **-, +**: Evaluation of the tolerance, see Page 62
- **@**: Transfer of an actual position, see Page 65

**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.

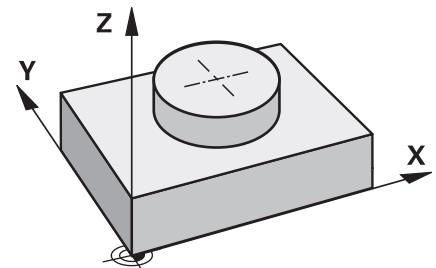
If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

**Further information:** "Cycle 1493 EXTRUSION PROBING", Page 303

### 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 47
- 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 136



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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from the first circle center</p>
Q973	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation starting from Diameter 1</p>

## Notes

### NOTICE

#### Danger of collision!

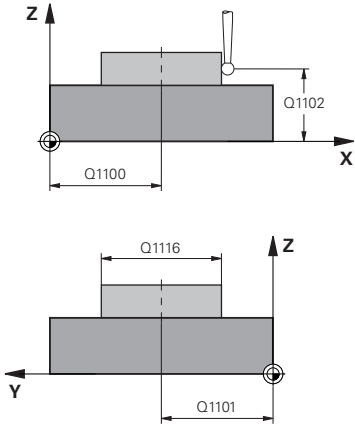
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ 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 57
- **"...-...+..."**: Evaluation of the tolerance, see Page 62
- **"...@..."**: Transfer of an actual position, see Page 65

#### 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 62

#### 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?

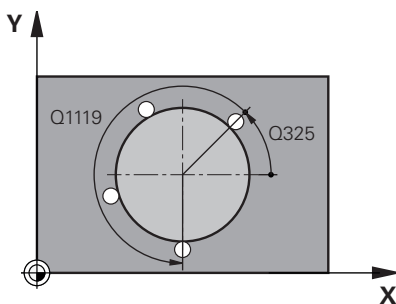
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

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

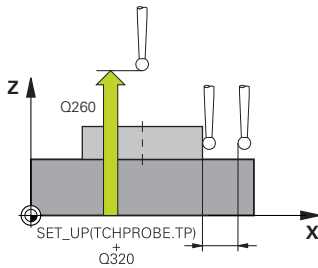
#### Q1119 Arc angular length?

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

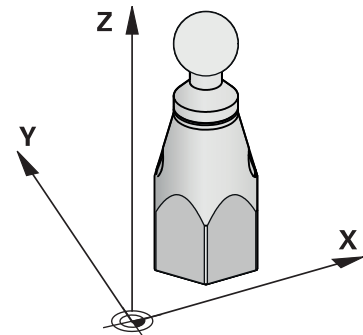
#### G1402

### 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 47
- 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 136



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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>

## Notes

### NOTICE

#### Danger of collision!

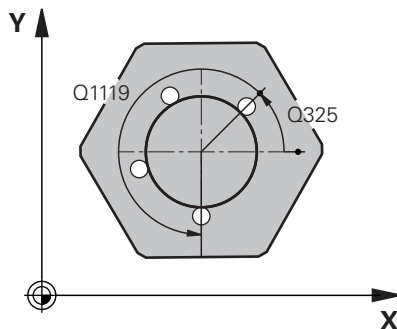
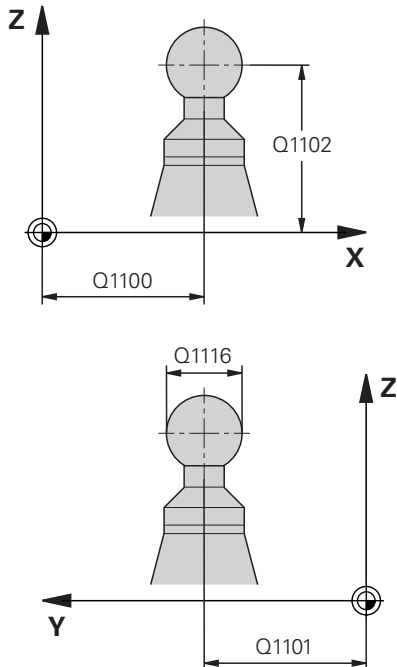
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ 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 57
- **"...-...+..."**: Evaluation of the tolerance, see Page 62
- **"...@..."**: Transfer of an actual position, see Page 65

#### 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 62

#### 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 303

### 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 47

- 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**:

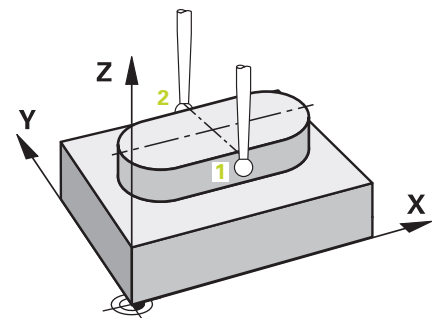
- If you program **CLEAR. HEIGHT MODE Q1125** with the value **0, 1** or **2**, the control positions the touch probe at **FMAX\_PROBE** back to **Q260 CLEARANCE HEIGHT**.

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 136



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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation from the center of the slot or ridge</p>
Q975	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</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.

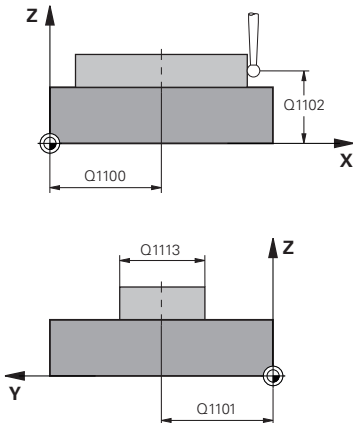
- ▶ Reset any coordinate transformations before the cycle call.

- 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 57
- **"...-...+..."**: Evaluation of the tolerance, see Page 62
- **"...@..."**: Transfer of an actual position, see Page 65

#### 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 62

#### 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?

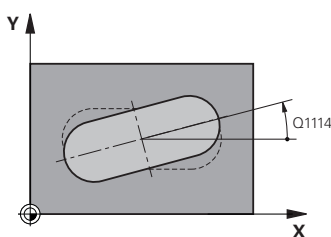
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET\_UP** column in the touch probe table. This value has an incremental effect.

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

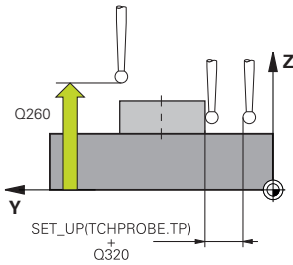
#### Q260 Clearance height?

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 303

**Cycle run**

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

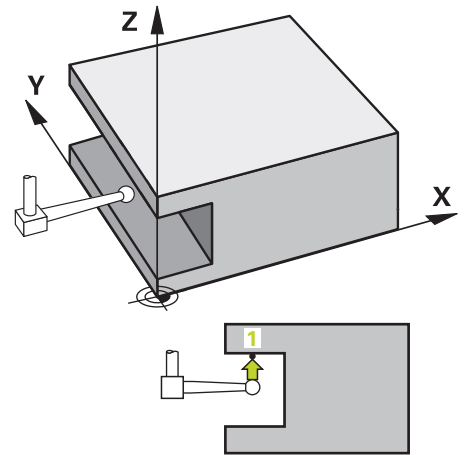
Pre-position in the machining plane based on the probing direction:

- **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 47

- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.
- 3 The control retracts the touch probe in the machining plane at **FMAX\_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
- 4 If you program **CLEAR. HEIGHT MODE Q1125** with the value **0, 1** or **2**, the control positions the touch probe at **FMAX\_PROBE** back to the clearance height **Q260**.
- 5 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

**Further information:** "Fundamentals of touch probe cycles 14xx for preset setting", Page 136



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"> <li>■ <b>-1</b> = Not defined</li> <li>■ <b>0</b> = Good</li> <li>■ <b>1</b> = Rework</li> <li>■ <b>2</b> = Scrap</li> <li>■ <b>3</b> = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation based on the nominal position of the first touch point</p>

## Notes

### NOTICE

#### Danger of collision!

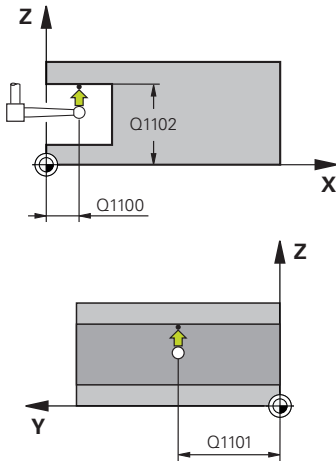
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ 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 137

## 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 57
- **-, +**: Evaluation of the tolerance, see Page 62
- **@**: Transfer of an actual position, see Page 65

#### 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.

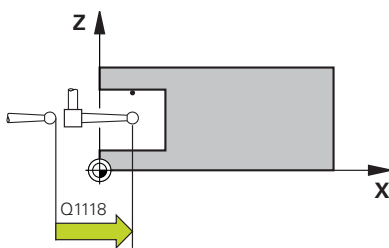
If **Q372 = +/- 1**: Distance is in the direction opposite to the probing direction.

If **Q372 = +/- 2**: Distance is in the direction opposite to the probing direction.

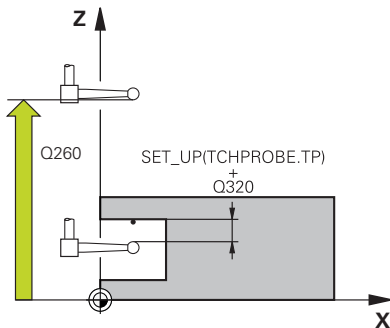
If **Q372 = +/- 3**: Distance is in the direction opposite to the angle of the spindle **Q336**.

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 303

**Cycle run**

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

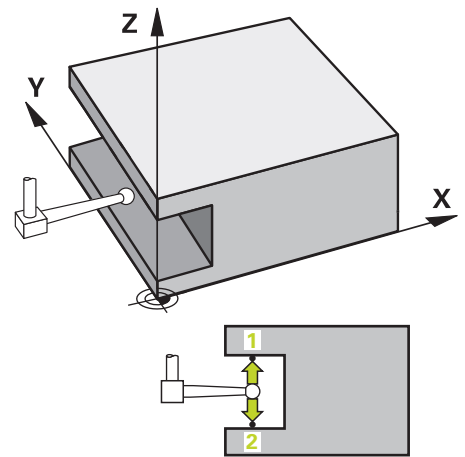
The pre-position in the machining plane depends on the object plane:

- **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 47

- 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 136



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"> <li>■ -1 = Not defined</li> <li>■ 0 = Good</li> <li>■ 1 = Rework</li> <li>■ 2 = Scrap</li> <li>■ 3 = Stylus not moved.</li> </ul> <p>The control displays the workpiece status <b>3</b> only in connection with the <b>441 FAST PROBING</b> cycle.</p> <p><b>Further information:</b> "Cycle 441 FAST PROBING", Page 300</p>
Q970	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation based on the center of the slot or the ridge</p>
Q975	<p>If you have programmed Cycle <b>1493 EXTRUSION PROBING</b>:</p> <p>Maximum deviation based on the slot width or ridge width</p>

## Notes

### NOTICE

#### Danger of collision!

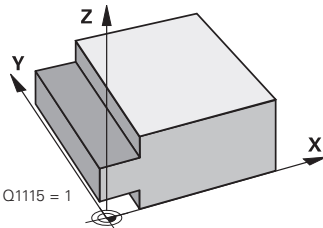
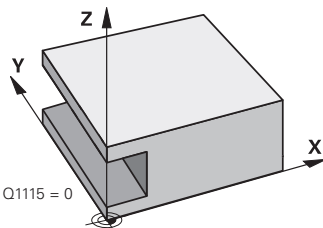
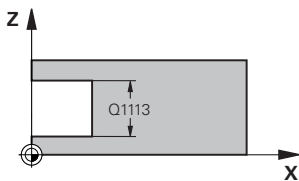
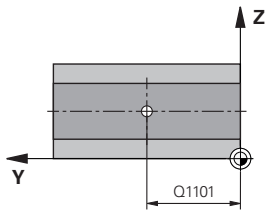
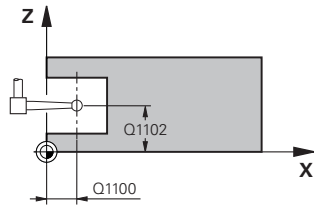
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

- ▶ 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 151

## 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 57
- **"...-...+..."**: Evaluation of the tolerance, see Page 62
- **"...@..."**: Transfer of an actual position, see Page 65

#### 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 62

#### Q1115 Geometry type (0/1)?

Type of object to be probed:

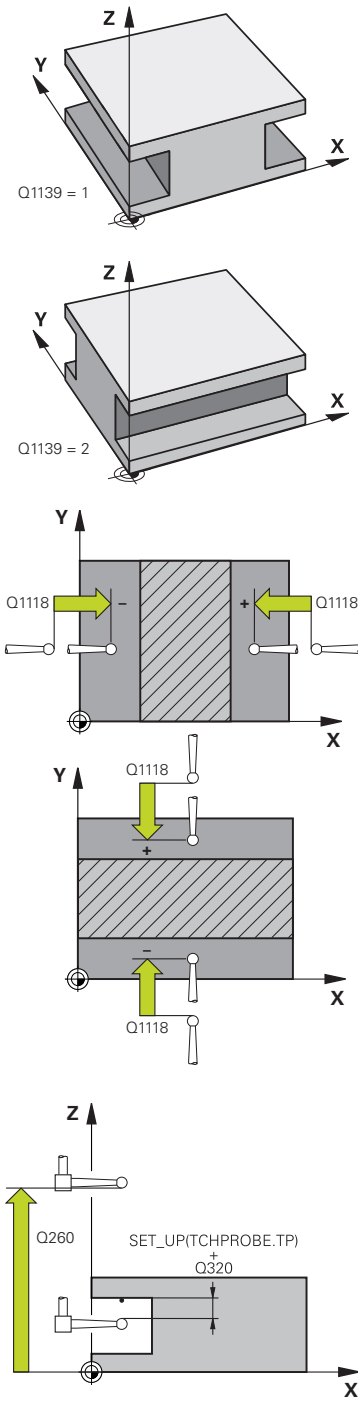
**0**: Slot

**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...9999.9999** 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: **-9999.9999...+9999.9999** 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

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

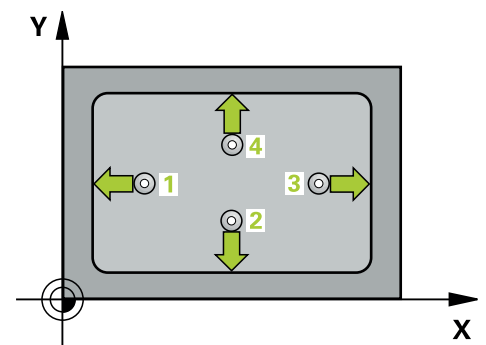
#### G410

### 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 47
- 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 167
- 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!

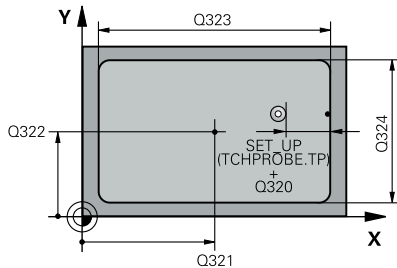
If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There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

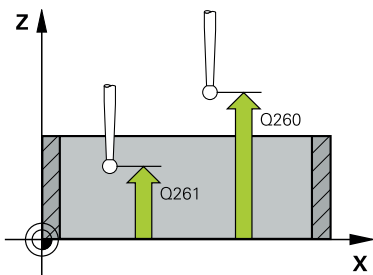
Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

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 167

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 167

**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

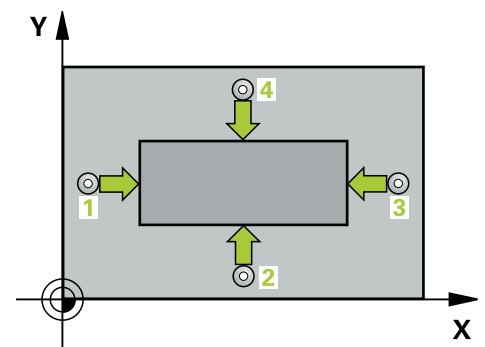
#### G411

### 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 47
- 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 167
- 7 Then the control saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.



Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

## Notes

### NOTICE

#### Danger of collision!

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

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

### NOTICE

#### Danger of collision!

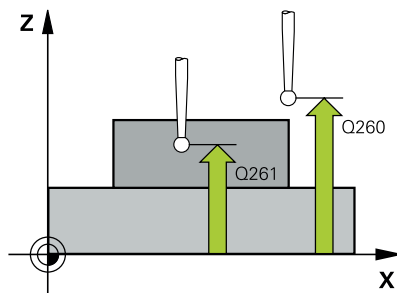
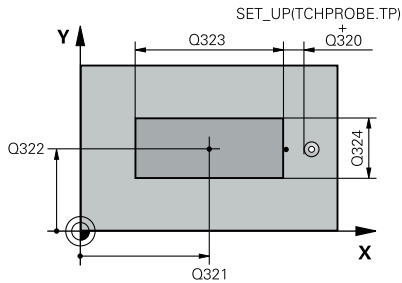
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

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

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

## Cycle parameters

### Help graphic



### Parameter

#### Q321 Center in 1st axis?

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

Input: **-99999.9999...+9999.9999**

#### Q322 Center in 2nd axis?

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

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

#### Q323 First side length?

Length of stud parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q324 Second side length?

Length of stud parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

#### Q261 Measuring height in probe axis?

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

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

#### Q320 Set-up clearance?

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

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

#### Q260 Clearance height?

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

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

#### Q301 Move to clearance height (0/1)?

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 167

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 167

**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**.

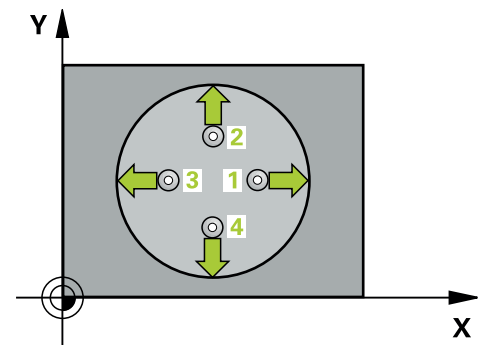
### Related topics

- Cycle **1401 CIRCLE PROBING**

**Further information:** "Cycle 1401 CIRCLE PROBING", Page 141

### 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 47
- 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 167
- 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

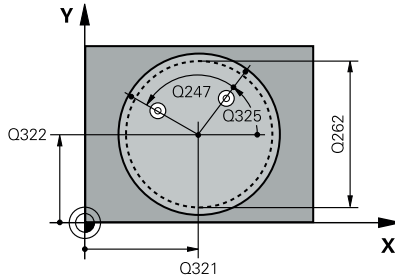
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°



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?

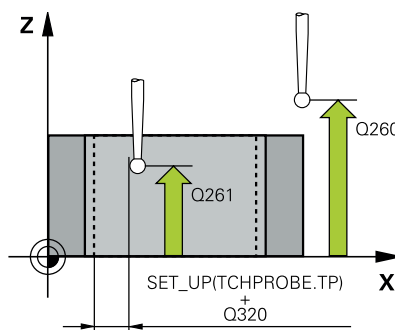
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET\_UP** column in the touch probe table. This value has an incremental effect.

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 167

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 167

**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**.

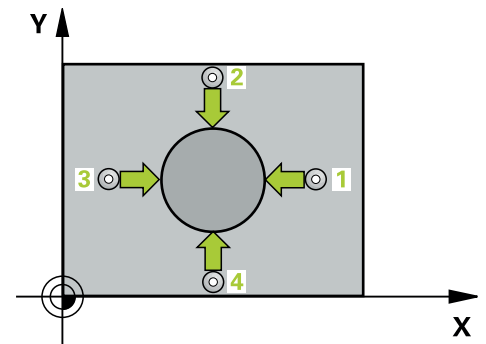
### Related topics

- Cycle **1401 CIRCLE PROBING**

**Further information:** "Cycle 1401 CIRCLE PROBING", Page 141

### 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 47
- 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 167
- 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.

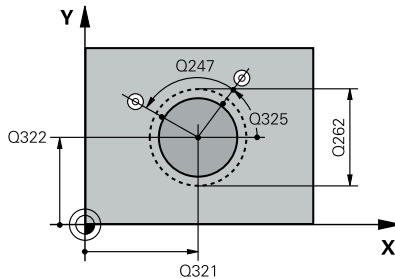
- ▶ Before a cycle definition you must have programmed a tool call to define the touch probe axis.
- The control will reset an active basic rotation at the beginning of the cycle.
  - This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
  - The smaller the stepping angle **Q247**, 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?

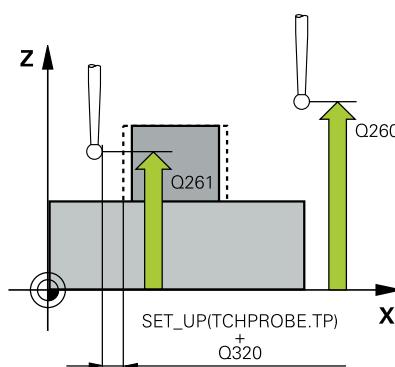
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET\_UP** column in the touch probe table. This value has an incremental effect.

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 167

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 167

**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

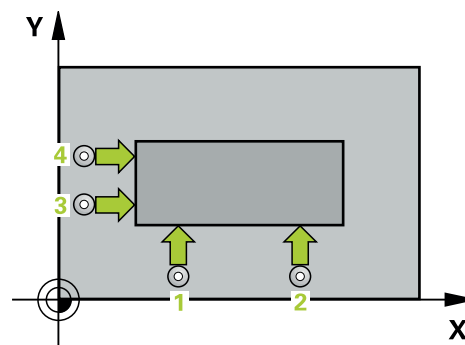
- Cycle **1416 INTERSECTION PROBING**

**Further information:** "Cycle 1416 INTERSECTION PROBING", Page 97

### 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 47
- 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 167
- 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
------	--

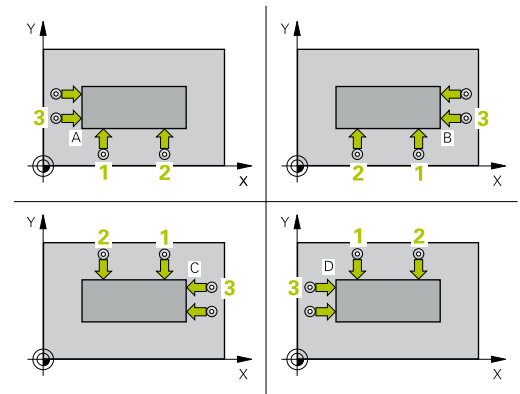
Q152	Actual value of corner in minor axis
------	--------------------------------------



**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 <b>1</b> greater than point <b>3</b>	Point <b>1</b> less than point <b>3</b>
B	Point <b>1</b> less than point <b>3</b>	Point <b>1</b> less than point <b>3</b>
C	Point <b>1</b> less than point <b>3</b>	Point <b>1</b> greater than point <b>3</b>
D	Point <b>1</b> greater than point <b>3</b>	Point <b>1</b> greater than point <b>3</b>

**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.

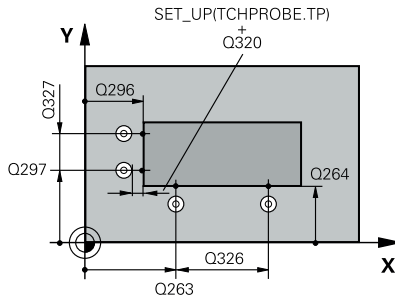
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

**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?

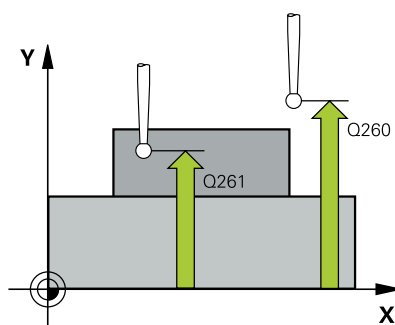
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

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 167

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 167

**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

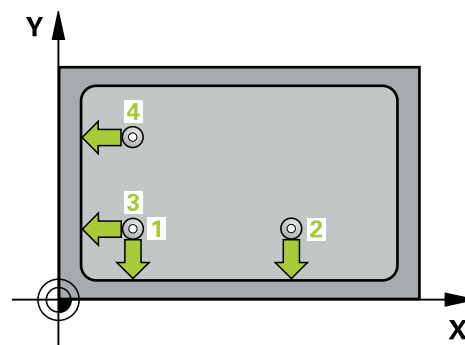
- Cycle **1416 INTERSECTION PROBING**

**Further information:** "Cycle 1416 INTERSECTION PROBING", Page 97

### 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 47
- 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 167
- 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.

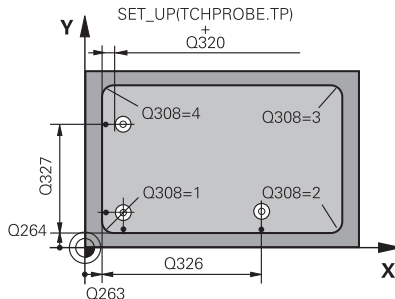
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

#### 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**

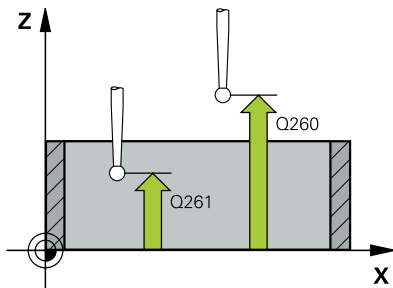
#### Q301 Move to clearance height (0/1)?

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 167

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 167

**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

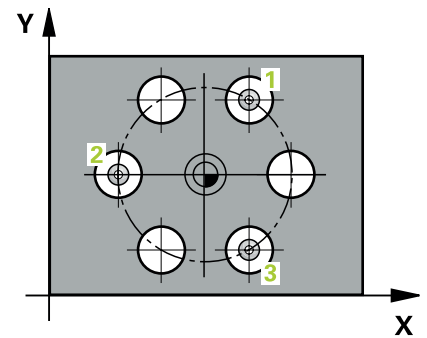
Touch probe cycle **416** finds the center of a bolt hole circle by measuring three holes, and defines the determined center as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

### 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 47

- 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 167
- 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.

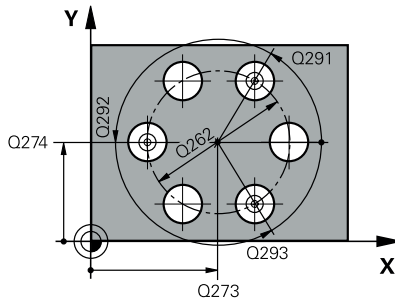
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

#### 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 167

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 167

**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**

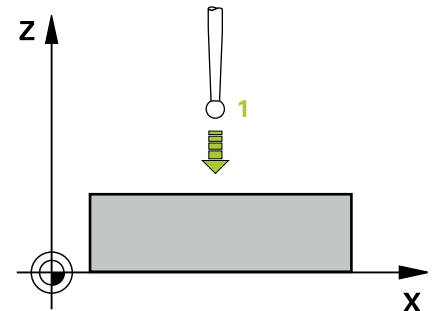
**Further information:** "Cycle 1400 POSITION PROBING", Page 137

### 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 47

- 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 167
- 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.

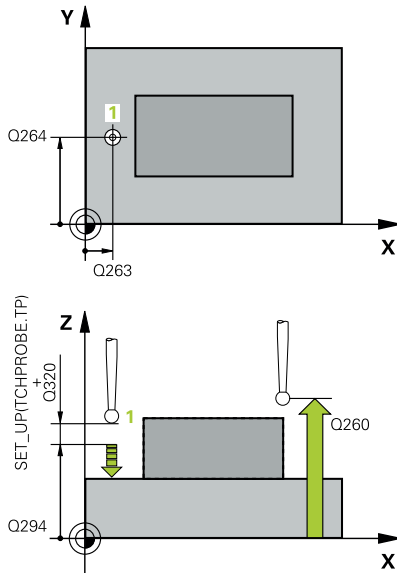
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control sets the preset in this axis.
- The control will reset an active basic rotation at the beginning of the cycle.

#### 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 167

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 167

**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

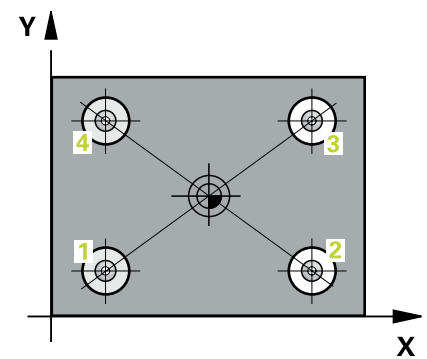
#### G418

### 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 47
- 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 167
- 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.

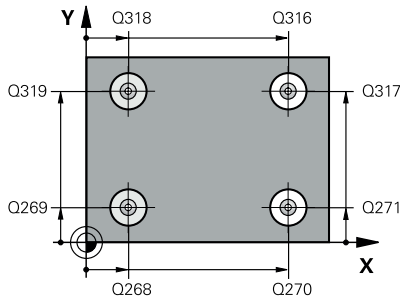
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

#### 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?

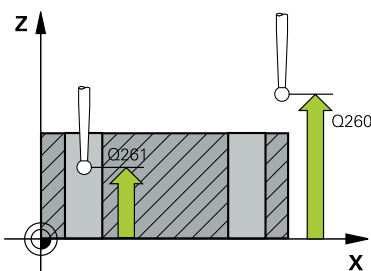
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

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 167

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 167

**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 137

### 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 47
- 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 167

## 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!

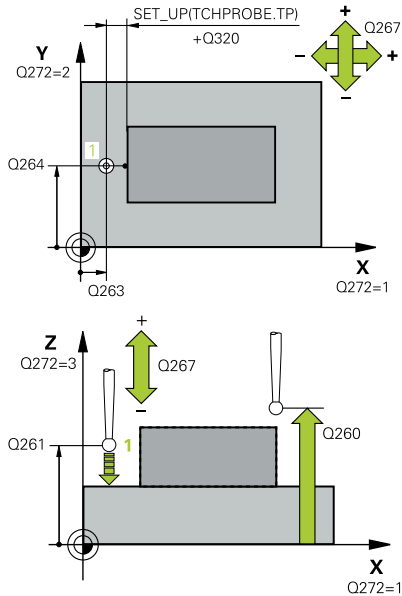
- ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
  - ▶ Reset any coordinate transformations beforehand.
- This cycle can only be executed 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 167

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 167

**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**.

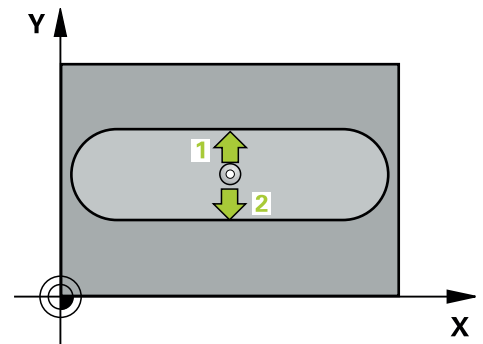
### Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

**Further information:** "Cycle 1404 PROBE SLOT/RIDGE", Page 151

### 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 47
- 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 167
- 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!

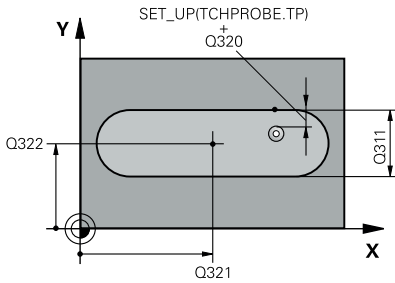
- ▶ To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width.
- ▶ Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- 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.

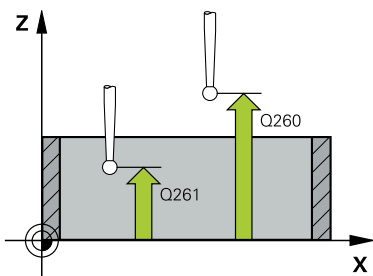
Input: **-99999.9999...+99999.9999** or **PREDEF**

#### 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 167

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**.

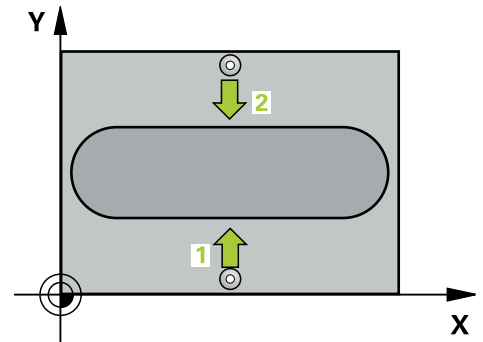
### Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

**Further information:** "Cycle 1404 PROBE SLOT/RIDGE", Page 151

### 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 47
- 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 167
- 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

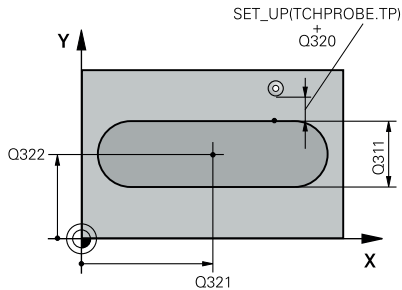
#### Danger of collision!

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?

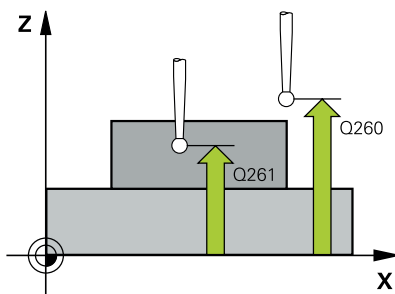
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET\_UP** column in the touch probe table. This value has an incremental effect.

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 167

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**

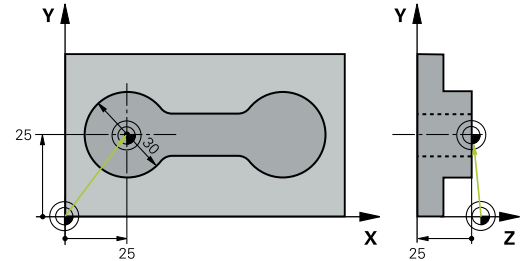
## Example

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

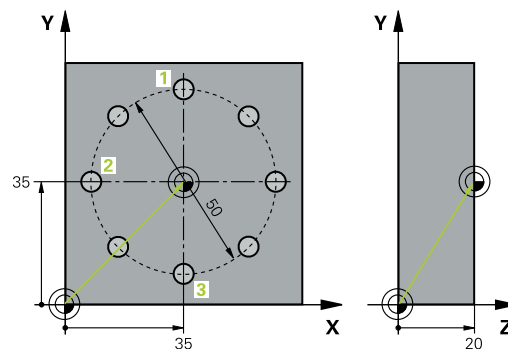


0 BEGIN PGM 413 MM	
1 TOOL CALL "TOUCH_PROBE" Z	
2 TCH PROBE 413 PRESET OUTS. CIRCLE ~	
Q321=+25	;CENTER IN 1ST AXIS ~
Q322=+25	;CENTER IN 2ND AXIS ~
Q262=+30	;NOMINAL DIAMETER ~
Q325=+90	;STARTING ANGLE ~
Q247=+45	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE 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"> <li>■ Measuring a coordinate in a selectable axis</li> </ul>	241
	Cycle 1 POLAR PRESET <ul style="list-style-type: none"> <li>■ Measuring a point</li> <li>■ Probing direction via angle</li> </ul>	243
	Cycle 420 MEASURE ANGLE <ul style="list-style-type: none"> <li>■ Measuring an angle in the working plane</li> </ul>	245
	Cycle 421 MEASURE HOLE <ul style="list-style-type: none"> <li>■ Measuring the position of a hole</li> <li>■ Measuring the diameter of a hole</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	248
	Cycle 422 MEAS. CIRCLE OUTSIDE <ul style="list-style-type: none"> <li>■ Measuring the position of a circular stud</li> <li>■ Measuring the diameter of a circular stud</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	253
	Cycle 423 MEAS. RECTAN. INSIDE <ul style="list-style-type: none"> <li>■ Measuring the position of a rectangular pocket</li> <li>■ Measuring the length and width of a rectangular pocket</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	258

Soft key	Cycle	Page
	Cycle 424 MEAS. RECTAN. OUTS. <ul style="list-style-type: none"> <li>■ Measuring the position of a rectangular stud</li> <li>■ Measuring the length and width of a rectangular stud</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	263
	Cycle 425 MEASURE INSIDE WIDTH <ul style="list-style-type: none"> <li>■ Measuring the position of a slot</li> <li>■ Measuring the width of a slot</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	267
	Cycle 426 MEASURE RIDGE WIDTH <ul style="list-style-type: none"> <li>■ Measuring the position of a ridge</li> <li>■ Measuring the width of a ridge</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	271
	Cycle 427 MEASURE COORDINATE <ul style="list-style-type: none"> <li>■ Measuring any coordinate in a selectable axis</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	275
	Cycle 430 MEAS. BOLT HOLE CIRC <ul style="list-style-type: none"> <li>■ Measuring the center point of a bolt hole circle</li> <li>■ Measuring the diameter of a bolt hole circle</li> <li>■ Nominal-to-actual value comparison, if applicable</li> </ul>	280
	Cycle 431 MEASURE PLANE <ul style="list-style-type: none"> <li>■ Finding the angle of a plane by measuring three points</li> </ul>	285

## 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

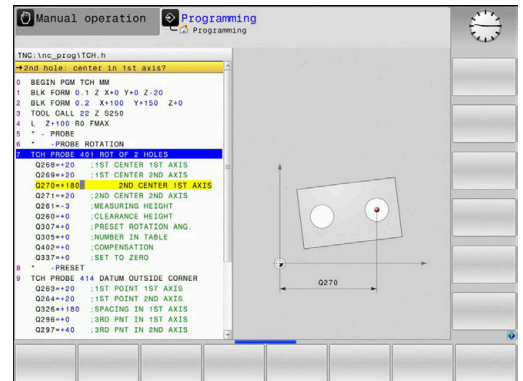
Further measuring results: Measuring height:	-5.0000
--	---------

**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
<b>Q180</b> = 1	Measurement results are within tolerance
<b>Q181</b> = 1	Rework is required
<b>Q182</b> = 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).

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**.

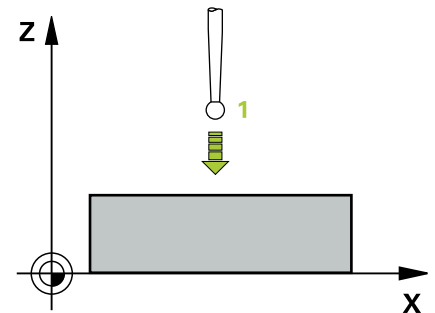
### Related topics

- Cycle **1400 POSITION PROBING**

**Further information:** "Cycle 1400 POSITION PROBING", Page 137

### 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><b>Parameter number for result?</b></p> <p>Enter the number of the Q parameter to which you want to assign the coordinate..</p> <p>Input: <b>0...1999</b></p>
	<p><b>Probing axis/probing direction?</b></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><b>Position value?</b></p> <p>Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe.</p> <p>Input: <b>-999999999...+999999999</b></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

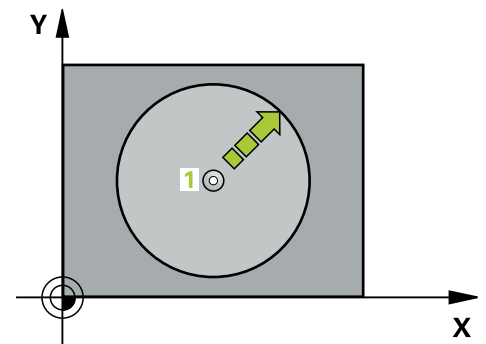
NC syntax is available only in Klartext programming.

### 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><b>Probing axis?</b></p> <p>Enter the probing axis with the axis key or the alphabetic keyboard. Confirm with the <b>ENT</b> key.</p> <p>Input: <b>X, Y, or Z</b></p>
	<p><b>Probing angle?</b></p> <p>Angle measured from the probing axis in which the touch probe will move.</p> <p>Input: <b>-180...+180</b></p>
	<p><b>Position value?</b></p> <p>Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe.</p> <p>Input: <b>-999999999...+999999999</b></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.



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 73

### 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 47
- 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:

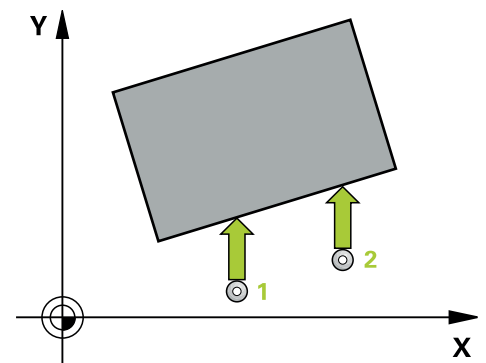
Q parameter number	Meaning
Q150	The measured angle is referenced to the main axis of the working plane.

### 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.

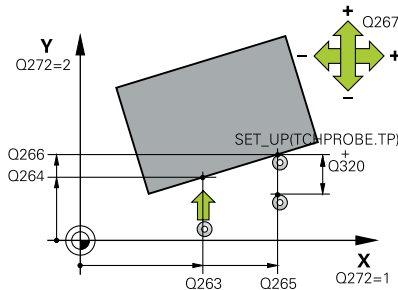
### Note on programming

- 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?

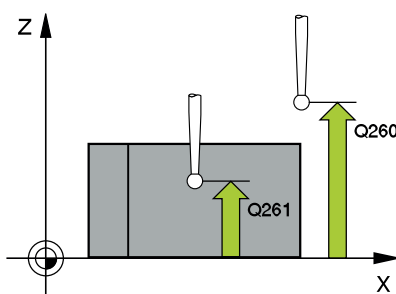
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

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**.

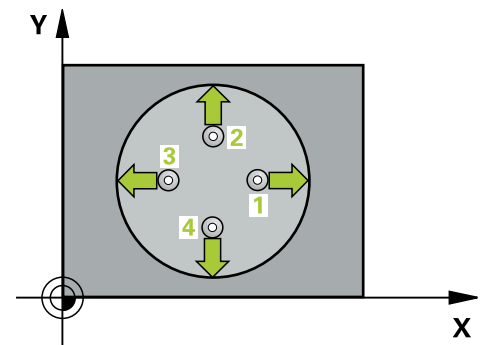
### Related topics

- Cycle **1401 CIRCLE PROBING**

**Further information:** "Cycle 1401 CIRCLE PROBING", Page 141

### 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 47
- 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

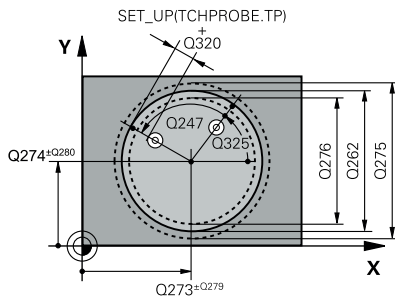
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

## 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**).
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

## Cycle parameters

### 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**

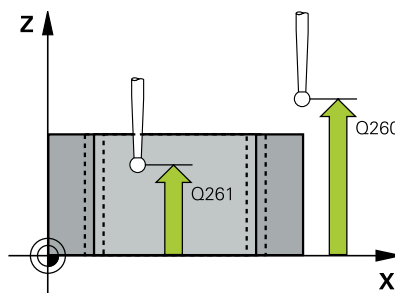
#### Q301 Move to clearance height (0/1)?

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 239

## 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**

Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

## 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**.

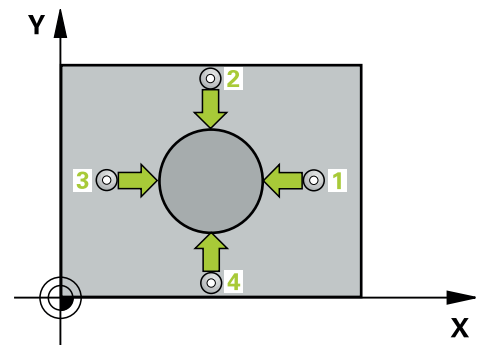
### Related topics

- Cycle **1401 CIRCLE PROBING**

**Further information:** "Cycle 1401 CIRCLE PROBING", Page 141

### 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 47
- 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

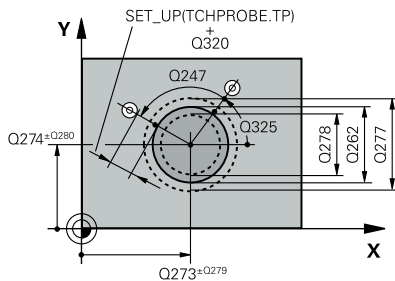
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

## Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

## Cycle parameters

### 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**

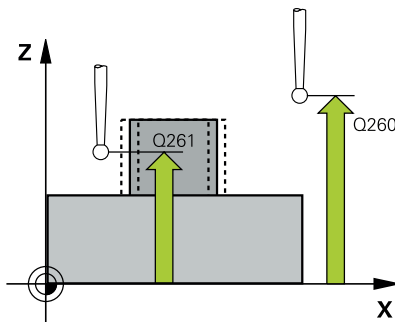
#### Q301 Move to clearance height (0/1)?

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 239

**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**

Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

## 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

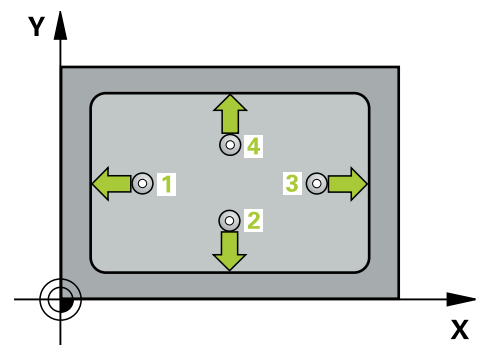
#### G423

### 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 47
- 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

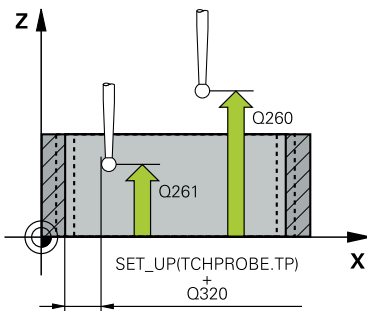
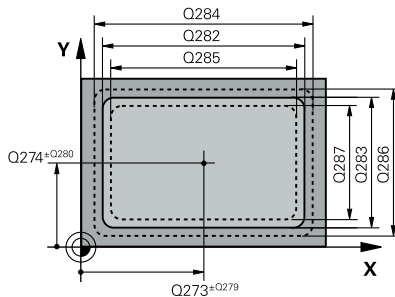
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe 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
	<p><b>Q286 Max. size limit 2nd side length?</b> Maximum permissible width for the pocket Input: <b>0...99999.9999</b></p>
	<p><b>Q287 Min. size limit 2nd side length?</b> Minimum permissible width for the pocket Input: <b>0...99999.9999</b></p>
	<p><b>Q279 Tolerance for center 1st axis?</b> Permissible position deviation in the main axis of the working plane. Input: <b>0...99999.9999</b></p>
	<p><b>Q280 Tolerance for center 2nd axis?</b> Permissible position deviation in the secondary axis of the working plane. Input: <b>0...99999.9999</b></p>
	<p><b>Q281 Measuring log (0/1/2)?</b> Define whether the control will create a measuring log: <b>0:</b> Do not create a measuring log. <b>1:</b> Create a measuring log: The control will save the <b>log file named TCHPR423.TXT</b> in the folder that also contains the associated NC program. <b>2:</b> Interrupt program run and display the measuring log on the control screen. Resume the NC program run with <b>NC Start</b>. Input: <b>0, 1, 2</b></p>
	<p><b>Q309 PGM stop if tolerance exceeded?</b> Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: <b>0:</b> Do not interrupt program run; no error message <b>1:</b> Interrupt program run and output an error message Input: <b>0, 1</b></p>
	<p><b>Q330 Tool for monitoring?</b> Define whether the control should perform tool monitoring: <b>0:</b> Monitoring not active <b>&gt; 0:</b> Tool number in tool table TOOL.T Input: <b>0...99999.9</b> or max. <b>255</b> characters <b>Further information:</b> "Tool monitoring", Page 239</p>

**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

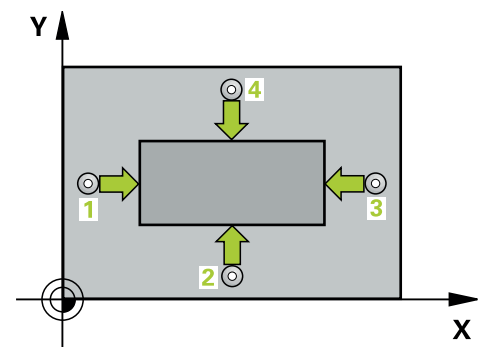
#### G424

### 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 47
- 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

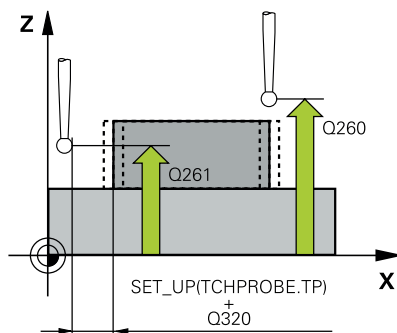
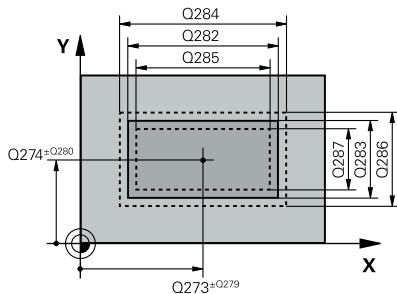
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

### 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
	<p><b>Q286 Max. size limit 2nd side length?</b> Maximum permissible width for the stud Input: <b>0...99999.9999</b></p>
	<p><b>Q287 Min. size limit 2nd side length?</b> Minimum permissible width for the stud Input: <b>0...99999.9999</b></p>
	<p><b>Q279 Tolerance for center 1st axis?</b> Permissible position deviation in the main axis of the working plane. Input: <b>0...99999.9999</b></p>
	<p><b>Q280 Tolerance for center 2nd axis?</b> Permissible position deviation in the secondary axis of the working plane. Input: <b>0...99999.9999</b></p>
	<p><b>Q281 Measuring log (0/1/2)?</b> Define whether the control will create a measuring log: <b>0:</b> Do not create a measuring log <b>1:</b> Create a measuring log: The control will save the <b>log file named TCHPR424.TXT</b> in the folder that also contains the .h file <b>2:</b> Interrupt program run and display the measuring log on the control screen. Resume the NC program run with <b>NC Start</b>. Input: <b>0, 1, 2</b></p>
	<p><b>Q309 PGM stop if tolerance exceeded?</b> Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: <b>0:</b> Do not interrupt program run; no error message <b>1:</b> Interrupt program run and output an error message Input: <b>0, 1</b></p>
	<p><b>Q330 Tool for monitoring?</b> Define whether the control should perform tool monitoring: <b>0:</b> Monitoring not active <b>&gt; 0:</b> 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: <b>0...99999.9</b> or max. <b>255</b> characters <b>Further information:</b> "Tool monitoring", Page 239</p>

**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 151

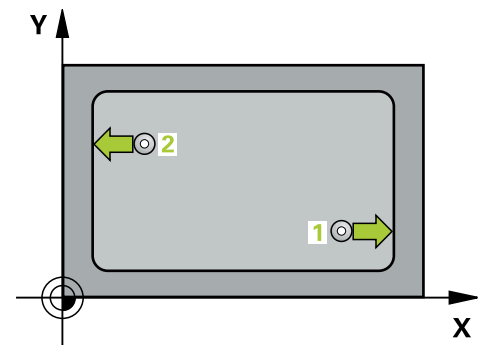
### Cycle run

- 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 47
- 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.

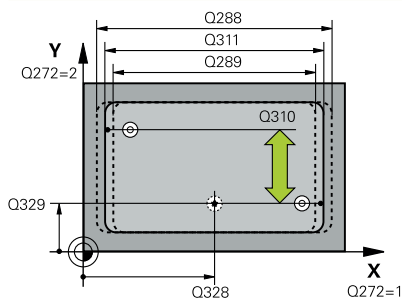


### Notes on programming

- 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?

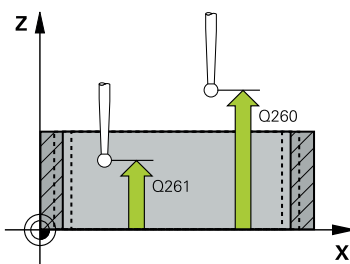
Maximum permissible length

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 239

**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 151

### 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 47
- 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:

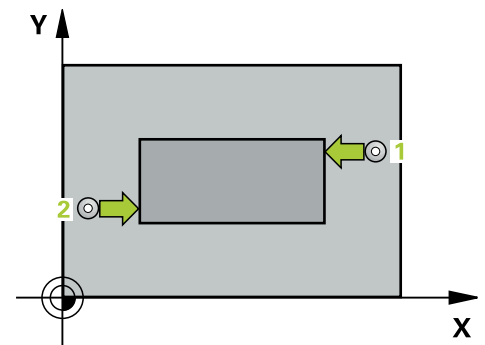
Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

### 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.

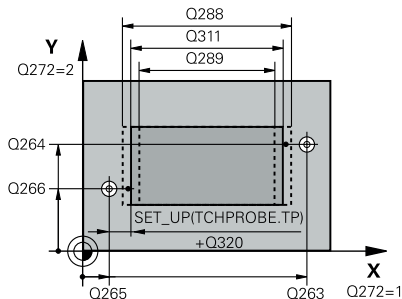
### Note on programming

- 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?

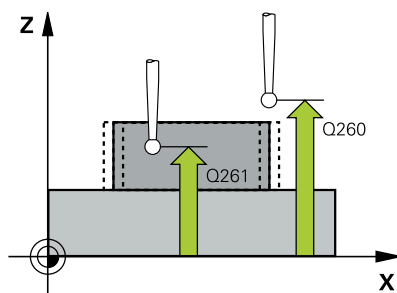
Nominal value of the length to be measured

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 239

**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.

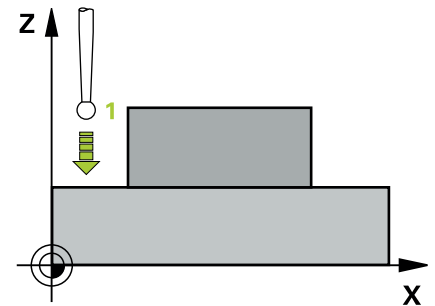
**i** Instead of Cycle **427 MEASURE COORDINATE**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

### Related topics

- Cycle **1400 POSITION PROBING**  
**Further information:** "Cycle 1400 POSITION PROBING", Page 137

### 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 47
- 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

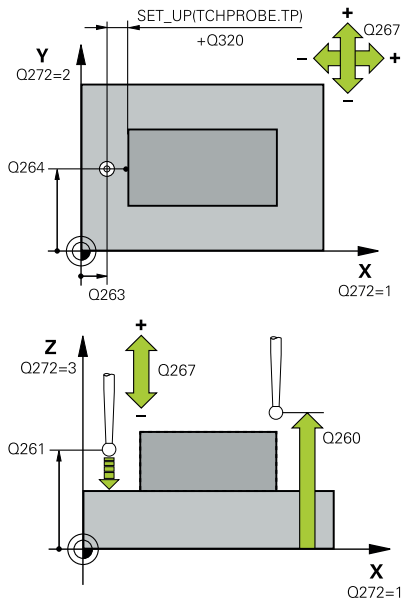
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing 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**).
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

## Cycle parameters

### 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 239

Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

**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

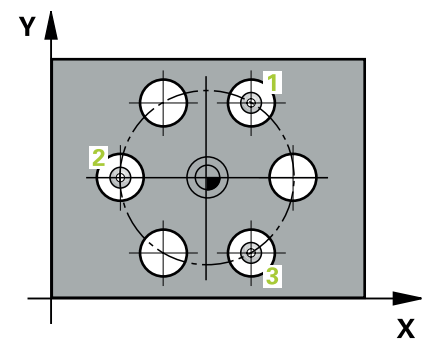
Touch probe cycle **430** finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

### 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 47

- 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

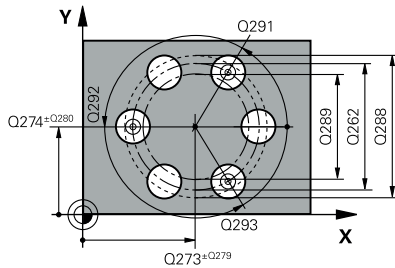
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **430** only monitors for tool breakage; there is no automatic tool compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

## 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?

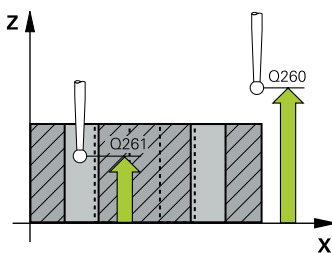
Minimum permissible diameter of bolt hole circle

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
	<p><b>Q280 Tolerance for center 2nd axis?</b> Permissible position deviation in the secondary axis of the working plane. Input: <b>0...99999.9999</b></p>
	<p><b>Q281 Measuring log (0/1/2)?</b> Define whether the control will create a measuring log: <b>0:</b> Do not create a measuring log <b>1:</b> Create a measuring log: The control will save the <b>log file named TCHPR430.TXT</b> in the folder that also contains the associated NC program <b>2:</b> Interrupt program run and display the measuring log on the control screen. Resume the NC program run with <b>NC Start</b>. Input: <b>0, 1, 2</b></p>
	<p><b>Q309 PGM stop if tolerance exceeded?</b> Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: <b>0:</b> Do not interrupt program run; no error message <b>1:</b> Interrupt program run and output an error message Input: <b>0, 1</b></p>
	<p><b>Q330 Tool for monitoring?</b> Define whether the control should perform tool monitoring: <b>0:</b> Monitoring not active <b>&gt; 0:</b> 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: <b>0...99999.9</b> or max. <b>255</b> characters <b>Further information:</b> "Tool monitoring", Page 239</p>

**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.



Instead of Cycle **431 MEASURE PLANE**, HEIDENHAIN recommends using the more powerful Cycle **1420 PROBING IN PLANE**.

### Related topics

- Cycle **1420 PROBING IN PLANE**

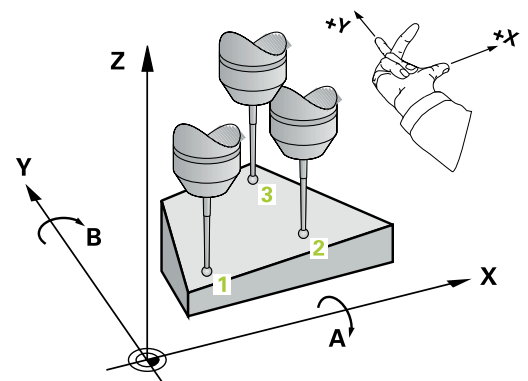
**Further information:** "Cycle 1420 PROBING IN PLANE", Page 66

### 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 47

- 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) -**

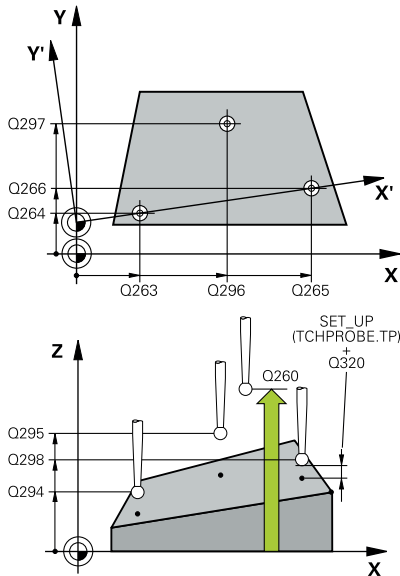
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three measuring points are not positioned on a straight line.
- The control will reset an active basic rotation at the beginning of the cycle.

#### 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

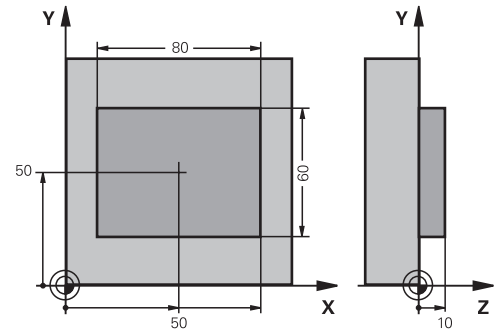


## 6.14 Programming examples

### Example: Measuring and reworking a rectangular stud

#### Program sequence

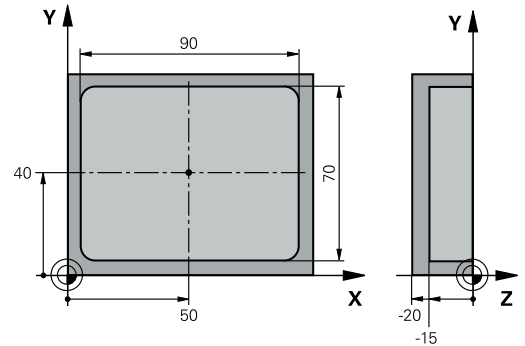
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



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"> <li>■ Touch probe cycle for defining OEM cycles</li> </ul>	295
	Cycle 4 MEASURING IN 3-D <ul style="list-style-type: none"> <li>■ Measuring any position</li> </ul>	298
	Cycle 441 FAST PROBING <ul style="list-style-type: none"> <li>■ Touch probe cycle for defining various touch probe parameters</li> </ul>	300
	Cycle 1493 EXTRUSION PROBING <ul style="list-style-type: none"> <li>■ Touch probe cycle for defining an extrusion</li> <li>■ Extrusion direction, length, and number of extrusion points can be programmed</li> </ul>	303

## 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
	<p><b>Parameter number for result?</b></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: <b>0...1999</b></p>
	<p><b>Probing axis?</b></p> <p>Enter the axis in whose direction the touch probe will move and confirm with the <b>ENT</b> key.</p> <p>Input: <b>X, Y, or Z</b></p>
	<p><b>Probing angle?</b></p> <p>This angle defines the probing direction. The angle refers to the probe axis. Confirm with the <b>ENT</b> key.</p> <p>Input: <b>-180...+180</b></p>
	<p><b>Maximum measuring range?</b></p> <p>Enter the maximum distance from the starting point by which the touch probe will move. Confirm with ENT.</p> <p>Input: <b>0...999999999</b></p>
	<p><b>Feed rate measurement</b></p> <p>Enter the measuring feed rate in mm/min.</p> <p>Input: <b>0...3000</b></p>
	<p><b>Maximum retraction distance?</b></p> <p>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.</p> <p>Input: <b>0...999999999</b></p>
	<p><b>Reference system? (0=ACT/1=REF)</b></p> <p>Define whether the probing direction and measurement result will be referenced to the current coordinate system (<b>ACT</b>, can be shifted or rotated) or the machine coordinate system (<b>REF</b>):</p> <p><b>0:</b> Perform the probing operation in the current system and save the measurement result in the <b>ACT</b> system</p> <p><b>1:</b> Perform the probing operation in the machine-based REF system. Save the measurement result in the REF system.</p> <p>Input: <b>0, 1</b></p>



---

**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><b>Parameter number for result?</b></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: <b>0...1999</b></p>
	<p><b>Relative measuring path in X?</b></p> <p>X component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: <b>-999999999...+999999999</b></p>
	<p><b>Relative measuring path in Y?</b></p> <p>Y component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: <b>-999999999...+999999999</b></p>
	<p><b>Relative measuring path in Z?</b></p> <p>Z component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: <b>-999999999...+999999999</b></p>
	<p><b>Maximum measuring range?</b></p> <p>Enter the maximum distance from the starting point by which the touch probe will move along the direction vector.</p> <p>Input: <b>-999999999...+999999999</b></p>
	<p><b>Feed rate measurement</b></p> <p>Enter the measuring feed rate in mm/min.</p> <p>Input: <b>0...3000</b></p>
	<p><b>Maximum retraction distance?</b></p> <p>Traverse path in the direction opposite the probing direction, after the stylus was deflected.</p> <p>Input: <b>0...999999999</b></p>
	<p><b>Reference system? (0=ACT/1=REF)</b></p> <p>Define whether the result of probing will be saved in the input coordinate system (<b>ACT</b>), or with respect to the machine coordinate system (<b>REF</b>):</p> <p><b>0:</b> Save the measurement result in the <b>ACT</b> system</p> <p><b>1:</b> Save the measurement result in the <b>REF</b> system</p> <p>Input: <b>0, 1</b></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 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><b>Q396 Positioning feed rate?</b></p> <p>Define the feed rate at which the touch probe will be moved to the specified positions.</p> <p>Input: <b>0...99999.999</b></p>
	<p><b>Q397 Pre-pos. at machine's rapid?</b></p> <p>Define whether the control, when repositioning the touch probe, traverses at <b>FMAX</b> feed rate (machine's rapid traverse):</p> <p><b>0:</b> Pre-position at the feed rate from <b>Q396</b></p> <p><b>1:</b> Pre-position at the machine's rapid traverse <b>FMAX</b></p> <p>Input: <b>0, 1</b></p>
	<p><b>Q399 Angle tracking (0/1)?</b></p> <p>Define whether the control will orient the touch probe before every probing operation:</p> <p><b>0:</b> Do not orient the spindle</p> <p><b>1:</b> Orient the spindle before every probing operation (increased accuracy)?</p> <p>Input: <b>0, 1</b></p>
	<p><b>Q400 Automatic interruption?</b></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><b>0:</b> 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><b>1:</b> Interrupt program run and output measurement results on the screen. You can then resume the NC program run with <b>NC Start</b>.</p> <p>Input: <b>0, 1</b></p> <p><b>Further information:</b> "Program interruption Q400=1", Page 300</p>
	<p><b>Q371 Touch point not reached?</b></p> <p>Define how the control behaves when the stylus does not move within the <b>DIST</b> value of the touch probe table.</p> <p><b>0:</b> The control interrupts the NC program with an error message saying that the touch point cannot be reached. This is standard behavior.</p> <p><b>1:</b> The control displays a warning and terminates the probing cycle. The NC program continues. Is effective only in the <b>14xx</b> cycles.</p> <p><b>2:</b> The control displays no warning and terminates the probing cycle. The NC program continues. Is effective only in the <b>14xx</b> cycles.</p> <p>Input: <b>0, 1, 2</b></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

## 7.5 Cycle 1493 EXTRUSION PROBING

### ISO programming

#### G1493

### Application

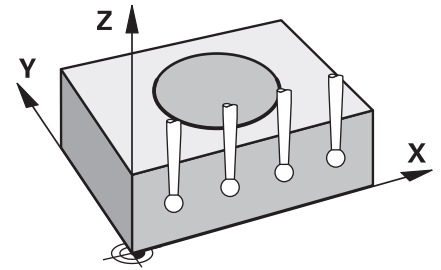
Cycle **1493** allows you to repeat the touch points of specific touch probe cycles along a straight line. In the cycle, you define the direction and the length of the extrusion, as well as the number of extrusion points.

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 66
- **PROBING ON EDGE** (Cycle **1410**, ISO: **G1410**), see Page 73
- **PROBING TWO CIRCLES** (Cycle **1411**, ISO: **G1411**), see Page 81
- **INCLINED EDGE PROBING** (Cycle **1412**, ISO: **G1412**), see Page 89
- **INTERSECTION PROBING** (Cycle **1416**, ISO: **G1416**), see Page 97
- **POSITION PROBING** (Cycle **1400**, ISO: **G1400**), see Page 137
- **CIRCLE PROBING** (Cycle **1401**, ISO: **G1401**), see Page 141
- **PROBE SLOT/RIDGE** (Cycle **1404**, ISO: **G1404**), see Page 151
- **PROBE POSITION OF UNDERCUT** (Cycle **1430**, ISO: **G1430**), see Page 155
- **PROBE SLOT/RIDGE UNDERCUT** (Cycle **1434**, ISO: **G1434**), see Page 161



**Result parameter Q**

The control saves the results of the touch probe cycle in the following Q parameters:

<b>Q parameter number</b>	<b>Meaning</b>
<b>Q970</b>	Maximum deviation from the ideal line of touch point 1
<b>Q971</b>	Maximum deviation from the ideal line of touch point 2
<b>Q972</b>	Maximum deviation from the ideal line of touch point 3
<b>Q973</b>	Maximum deviation of diameter 1
<b>Q974</b>	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
<b>QS970</b>	Results of touch point 1 of an extrusion
<b>QS971</b>	Results of touch point 2 of an extrusion
<b>QS972</b>	Results of touch point 3 of an extrusion
<b>QS973</b>	Results of diameter 1 of an extrusion
<b>QS974</b>	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.

<b>11 QS0 = SUBSTR ( SRC_QS970 BEG0 LEN10 )</b>	; Read out the first result from <b>QS970</b>
<b>12 QL1 = TONUMB ( SRC_QS0 )</b>	; Convert alphanumeric value from <b>QS0</b> to a numerical value and assign it to <b>QL0</b>
<b>13 QS0 = SUBSTR ( SRC_QS970 BEG11 LEN10 )</b>	; Read out the second result from <b>QS970</b>
<b>14 QL2 = TONUMB ( SRC_QS0 )</b>	; Convert alphanumeric value from <b>QS0</b> to a numerical value and assign it to <b>QL2</b>

**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:

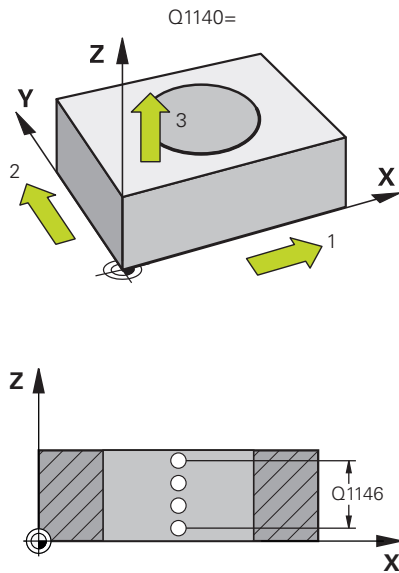
- Nominal position of touch point 1: 2.35 mm
- Results: **QS970** = 2.30000000 2.35000000 2.40000000  
2.50000000

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

## 7.6 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.

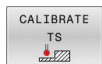
During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

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

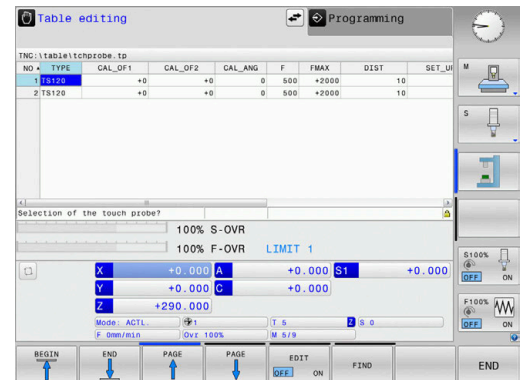
Calibration cycles provided by the control

Soft key	Function	Page
	Cycle 461 TS CALIBRATION OF TOOL LENGTH <ul style="list-style-type: none"> <li>■ Calibrating the length</li> </ul>	310
	Cycle 462 CALIBRATION OF A TS IN A RING <ul style="list-style-type: none"> <li>■ Measuring the radius using a ring gauge</li> <li>■ Measuring the center offset using a ring gauge</li> </ul>	312
	Cycle 463 TS CALIBRATION ON STUD <ul style="list-style-type: none"> <li>■ Measuring the radius using a stud or a calibration pin</li> <li>■ Measuring the center offset using a stud or a calibration pin</li> </ul>	315
	Cycle 460 CALIBRATION OF TS ON A SPHERE <ul style="list-style-type: none"> <li>■ Measuring the radius using a calibration sphere</li> <li>■ Measuring the center offset using a calibration sphere</li> </ul>	318

## 7.7 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**

## 7.8 Cycle 461 TS CALIBRATION OF TOOL LENGTH

### ISO programming

G461

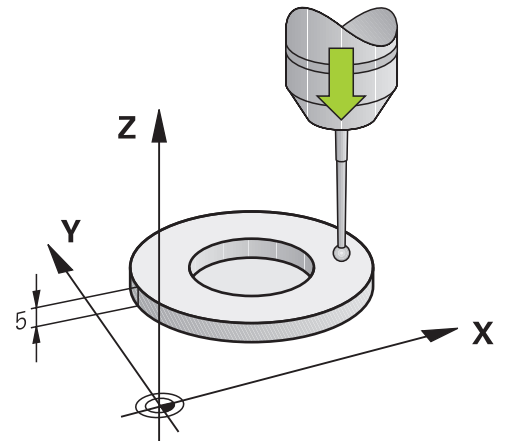
### Application



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!**

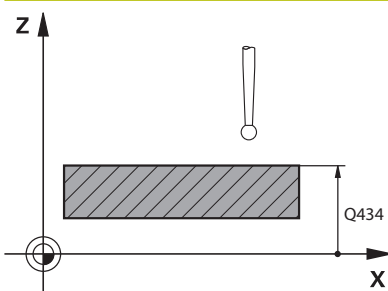
When running touch probe cycles **400 to 499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

- ▶ 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**

## 7.9 Cycle 462 CALIBRATION OF A TS IN A RING

### ISO programming

#### G462

#### Application



Refer to your machine manual.

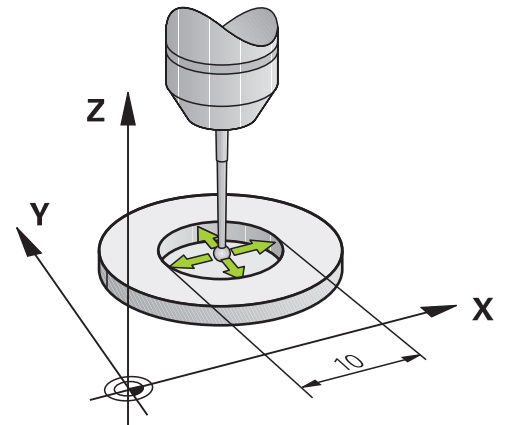
Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

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.

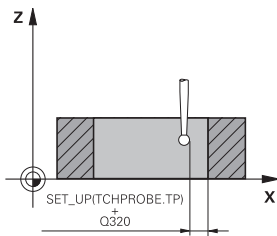
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

**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

## 7.10 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.

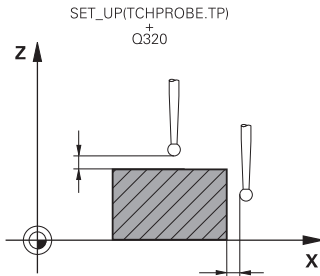
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

### 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

## 7.11 Cycle 460 CALIBRATION OF TS ON A SPHERE

### ISO programming

G460

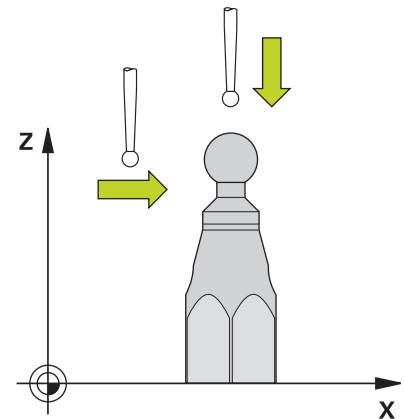
### Application



Refer to your machine manual.

With Cycle **460** you can calibrate a triggering 3-D touch probe automatically using an exact calibration sphere.

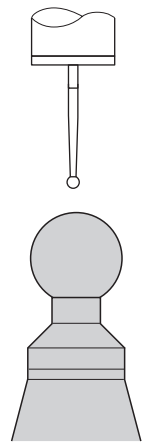
It is also possible to capture 3D calibration data. Option 92, **3D-ToolComp**, is required for this purpose. 3D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3D calibration data are stored under TNC:\system\3D-ToolComp\\*. The **DR2TABLE** column 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

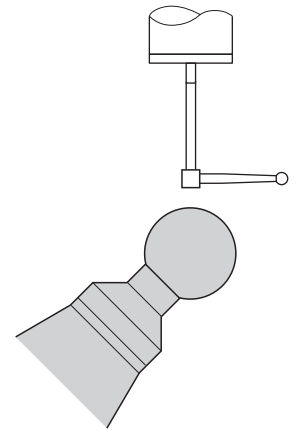
**Before calibrating with an L-shaped stylus:**

- ▶ Clamp the calibration sphere

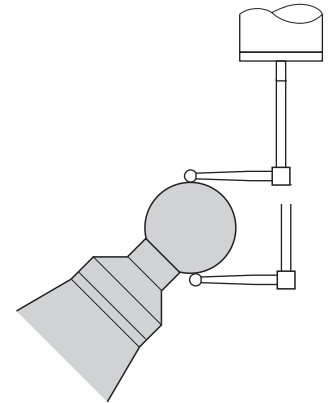


It must be possible to probe the north pole and south pole during calibration. If this is not possible, the control cannot determine the sphere radius. Ensure that no collision can occur.

- ▶ 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



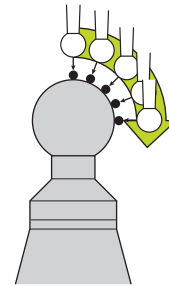
Calibration process with an L-shaped stylus

**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.

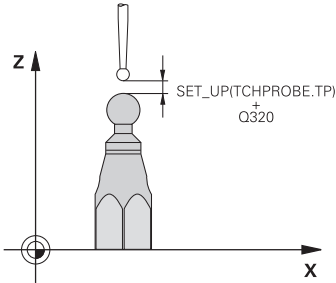
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be 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 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 Tool  
Measurement**

## 8.1 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 ■ Calibrating the tool touch probe	333
		Cycle 31 or 481 CAL. TOOL LENGTH ■ Measuring the tool length	336
		Cycle 32 or 482 CAL. TOOL RADIUS ■ Measuring the tool radius	340
		Cycle 33 or 483 MEASURE TOOL ■ Measuring the tool length and radius	345
		Cycle 484 CALIBRATE IR TT ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	350



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** 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
<b>n</b>	Shaft speed [rpm]
<b>maxPeriphSpeedMeas</b>	Maximum permissible cutting speed in m/min
<b>r</b>	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
<b>v</b>	Probing feed rate [mm/min]
<b>Measuring tolerance</b>	Measuring tolerance [mm], depending on <b>maxPeriphSpeedMeas</b>
<b>n</b>	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	<b>measureTolerance1</b>
30 to 60 mm	<b>2 • measureTolerance1</b>
60 to 90 mm	<b>3 • measureTolerance1</b>
90 to 120 mm	<b>4 • measureTolerance1</b>

**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
<b>r</b>	Active tool radius [mm]
<b>measureTolerance1</b>	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 tools

Abbr.	Inputs	Dialog
<b>CUT</b>	The number of teeth of the tool for automatic tool measurement or cutting data calculation (maximum of 20 teeth)	<b>Number of teeth?</b>
<b>LTOL</b>	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 <b>TL</b> (status <b>L</b> ). Input: <b>0.0000...5.0000</b>	<b>Wear tolerance: length?</b>
<b>RTOL</b>	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 <b>TL</b> (status <b>L</b> ). Input: <b>0.0000...5.0000</b>	<b>Wear tolerance: radius?</b>
<b>DIRECT.</b>	Cutting direction of the tool for automatic tool measurement with a rotating tool. Input: -, +	<b>Cutting direction (M3 = -)?</b>
<b>R-OFFS</b>	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: <b>-99999.9999...+99999.9999</b>	<b>Tool offset: radius?</b>
<b>L-OFFS</b>	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 <b>offsetToolAxis</b> machine parameter (no. 122707). Input: <b>-99999.9999...+99999.9999</b>	<b>Tool offset: length?</b>
<b>LBREAK</b>	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 <b>TL</b> (status <b>L</b> ). Input: <b>0.0000...9.0000</b>	<b>Breakage tolerance: length?</b>
<b>RBREAK</b>	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 <b>TL</b> (status <b>L</b> ). Input: <b>0.0000...9.0000</b>	<b>Breakage tolerance: radius?</b>

## Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
<b>Drill</b>	No function	0: No offset required because tool tip is to be measured	
<b>End mill</b>	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 <b>offsetToolAxis</b> (no. 122707) used.
<b>Spherical cutter</b> 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.

## 8.2 Cycle 30 or 480 CALIBRATE TT

### ISO programming

G480

### Application



Refer to your machine manual!

You calibrate the TT with touch probe cycle **30** or **480** (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 327). 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><b>Q260 Clearance height?</b></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 <b>safetyDistToolAx</b> (no. 114203)).</p> <p>Input: <b>-99999.9999...+99999.9999</b></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
```

## 8.3 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 327). 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.

## Cycle parameters

Help graphic	Parameter
	<p><b>Q340 Tool measurement mode (0-2)?</b></p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p><b>0:</b> 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.</p> <p><b>1:</b> 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 <b>Q115</b>. 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).</p> <p><b>2:</b> 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 <b>Q115</b>. Nothing is entered under L or DL in the tool table.</p> <p>Input: <b>0, 1, 2</b></p>
	<p><b>Q260 Clearance height?</b></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 <b>safetyDistStylus</b>).</p> <p>Input: <b>-99999.9999...+99999.9999</b></p>
	<p><b>Q341 Probe the teeth? 0=no/1=yes</b></p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: <b>0, 1</b></p>

### 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><b>Parameter number for result?</b></p> <p>Parameter number in which the control stores the status of the measurement:</p> <p><b>0.0:</b> Tool is within the tolerance</p> <p><b>1.0:</b> Tool is worn (<b>LTOL</b> exceeded)</p> <p><b>2.0:</b> Tool is broken (<b>LBREAK</b> exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with <b>NO ENT</b></p> <p>Input: <b>0...1999</b></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
```

## 8.4 Cycle 32 or 482 CAL. TOOL RADIUS

### ISO programming

G482

### Application



Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **32** or **482** (see "Differences between Cycles 31 to 33 and Cycles 481 to 483", Page 327). 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 342

### 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.

**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:

$$\varepsilon = 90 - \operatorname{atan} \left( \frac{h_{[tt]}}{\frac{R \times 2 \times \pi}{x}} \right)$$

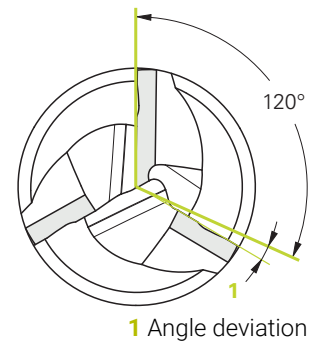
Abbreviation	Definition
$\varepsilon$	Maximum angle of twist
$h_{[tt]}$	Height of tool touch probe contact
<b>R</b>	Tool radius
<b>x</b>	Number of teeth of tool

**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><b>Q340 Tool measurement mode (0-2)?</b></p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p><b>0:</b> 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><b>1:</b> 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 <b>Q116</b>. 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><b>2:</b> 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 <b>Q116</b>. Nothing is entered under R or DR in the tool table.</p> <p>Input: <b>0, 1, 2</b></p>
	<p><b>Q260 Clearance height?</b></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 <b>safetyDistStylus</b>).</p> <p>Input: <b>-99999.9999...+99999.9999</b></p>
	<p><b>Q341 Probe the teeth? 0=no/1=yes</b></p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: <b>0, 1</b></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><b>Parameter number for result?</b></p> <p>Parameter number in which the control stores the status of the measurement:</p> <p><b>0.0:</b> Tool is within the tolerance</p> <p><b>1.0:</b> Tool is worn (<b>RTOL</b> exceeded)</p> <p><b>2.0:</b> Tool is broken (<b>RBREAK</b> exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with <b>NO ENT</b></p> <p>Input: <b>0...1999</b></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
```



## 8.5 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 327). 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 347

## 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.

#### 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.

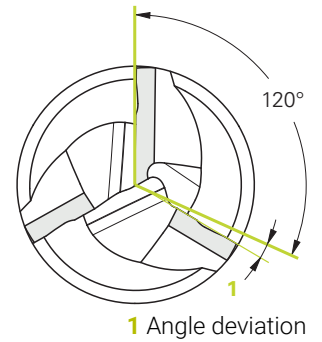
**Notes for individual tooth measurement of radius Q341=1**

**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:

$$\epsilon = 90 - \text{atan} (h[\text{tt}] / (\text{tool radius} * 2 * \pi / \text{number of teeth}))$$

Abbreviation	Definition
$\epsilon$	Maximum angle of twist
$h[\text{tt}]$	Height of tool touch probe contact

**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><b>Q340 Tool measurement mode (0-2)?</b></p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p><b>0:</b> 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><b>1:</b> 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 <b>Q115</b> and <b>Q116</b>. 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><b>2:</b> 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 <b>Q115</b> or <b>Q116</b>. Nothing is entered under L, R, or DL, DR in the tool table.</p> <p>Input: <b>0, 1, 2</b></p>
	<p><b>Q260 Clearance height?</b></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 <b>safetyDistStylus</b>).</p> <p>Input: <b>-99999.9999...+99999.9999</b></p>
	<p><b>Q341 Probe the teeth? 0=no/1=yes</b></p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: <b>0, 1</b></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><b>Parameter number for result?</b></p> <p>Parameter number in which the control stores the status of the measurement:</p> <p><b>0.0:</b> Tool is within the tolerance</p> <p><b>1.0:</b> Tool is worn (<b>LTOL</b> or/and <b>RTOL</b> exceeded)</p> <p><b>2.0:</b> Tool is broken (<b>LBREAK</b> or/and <b>RBREAK</b> exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with <b>NO ENT</b>.</p> <p>Input: <b>0...1999</b></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
```

## 8.6 Cycle 484 CALIBRATE IR TT

### ISO programming

#### G484

### Application

Cycle **484** allows you to calibrate your tool touch probe (e.g., the wireless infrared TT 460 tool touch probe). You can perform the calibration process with or without manual intervention.

- **With manual intervention:** If you define **Q536** = 0, then the control will stop before the calibration process. You then need to position the calibration tool manually above the center of the tool touch probe.
- **Without manual intervention:** If you define **Q536** = 1, then the control will automatically execute the cycle. You may have to program a 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><b>Q536 Stop before running (0=Stop)?</b></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><b>0:</b> 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 <b>NC Start</b> to continue the calibration process or press the <b>CANCEL</b> soft key to cancel the calibration process.</p> <p><b>1:</b> Without stopping before the calibration process. The control starts the calibration process depending on <b>Q523</b>. Before running Cycle <b>484</b>, you may have to position the tool above the tool touch probe.</p> <p>Input: <b>0, 1</b></p>
	<p><b>Q523 Position of tool probe (0-2)?</b></p> <p>Position of the tool touch probe:</p> <p><b>0:</b> Current position of the calibration tool. The tool touch probe is below the current position of the calibration tool. If <b>Q536 = 0</b>, position the calibration tool manually above the center of the tool touch probe during the cycle. If <b>Q536 = 1</b>, you need to position the calibration tool above the center of the tool touch probe before the start of the cycle.</p> <p><b>1:</b> Configured position of the tool touch probe. The control adopts the position from the machine parameter <b>centerPos</b> (no. 114201). You do not need to pre-position the tool. The calibration tool approaches the position automatically.</p> <p><b>2:</b> Current position of the calibration tool. See <b>Q523 = 0. 0</b>. The control additionally writes the determined position (where applicable) to the machine parameter <b>centerPos</b> (no. 114201) after calibration.</p> <p>Input: <b>0, 1, 2</b></p>

### Example

11 TOOL CALL 12 Z	
12 TCH PROBE 484 CALIBRATE IR TT ~	
Q536=+0	;STOP BEFORE RUNNING ~
Q523=+0	;TT POSITION





# 9








**Cycles: Special  
Functions**

## 9.1 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
	<b>9 DWELL TIME</b> <ul style="list-style-type: none"> <li>■ Delay execution by the programmed dwell time</li> </ul>	<b>Further information:</b> User's Manual for <b>Programming of Machining Cycles</b>
	<b>12 PGM CALL</b> <ul style="list-style-type: none"> <li>■ Call any NC program</li> </ul>	<b>Further information:</b> User's Manual for <b>Programming of Machining Cycles</b>
	<b>13 ORIENTATION</b> <ul style="list-style-type: none"> <li>■ Rotate spindle to a specific angle</li> </ul>	357
	<b>32 TOLERANCE</b> <ul style="list-style-type: none"> <li>■ Program the permissible contour deviation for jerk-free machining operations</li> </ul>	<b>Further information:</b> User's Manual for <b>Programming of Machining Cycles</b>
	<b>225 ENGRAVING</b> <ul style="list-style-type: none"> <li>■ Engrave texts on a plane surface</li> <li>■ Arranged in a straight line or along a circular arc</li> </ul>	<b>Further information:</b> User's Manual for <b>Programming of Machining Cycles</b>
	<b>232 FACE MILLING</b> <ul style="list-style-type: none"> <li>■ Face mill a level surface in multiple infeeds</li> <li>■ Selection of the milling plan</li> </ul>	<b>Further information:</b> User's Manual for <b>Programming of Machining Cycles</b>
	<b>18 THREAD CUTTING</b> <ul style="list-style-type: none"> <li>■ With controlled spindle</li> <li>■ Spindle stops at the bottom of the hole</li> </ul>	<b>Further information:</b> User's Manual for <b>Programming of Machining Cycles</b>

## 9.2 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** machining mode.

### Cycle parameters

#### Help graphic

#### Parameter

##### Orientation angle

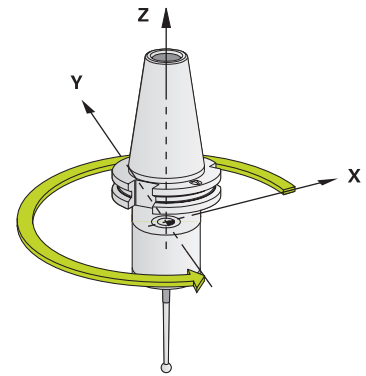
Enter the angle relative to the angle reference axis of the working plane.

Input: **0...360**

### Example

11 CYCL DEF 13.0 ORIENTATION

12 CYCL DEF 13.1 ANGLE180





# 10

**Tables of Cycles**

## 10.1 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:  
1303429-xx

### Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	REF. PLANE	■		241
1	POLAR PRESET	■		243
3	MEASURING	■		295
4	MEASURING IN 3-D	■		298
30	CALIBRATE TT	■		333
31	CAL. TOOL LENGTH	■		336
32	CAL. TOOL RADIUS	■		340
33	MEASURE TOOL	■		345
400	BASIC ROTATION	■		107
401	ROT OF 2 HOLES	■		111
402	ROT OF 2 STUDS	■		116
403	ROT IN ROTARY AXIS	■		121
404	SET BASIC ROTATION	■		130
405	ROT IN C AXIS	■		126
408	SLOT CENTER PRESET	■		223
409	RIDGE CENTER PRESET	■		228
410	PRESET INSIDE RECTAN	■		169
411	PRESET OUTS. RECTAN	■		174
412	PRESET INSIDE CIRCLE	■		180
413	PRESET OUTS. CIRCLE	■		186
414	PRESET OUTS. CORNER	■		192
415	PRESET INSIDE CORNER	■		198
416	PRESET CIRCLE CENTER	■		204
417	PRESET IN TS AXIS	■		210
418	PRESET FROM 4 HOLES	■		214
419	PRESET IN ONE AXIS	■		219
420	MEASURE ANGLE	■		245
421	MEASURE HOLE	■		248
422	MEAS. CIRCLE OUTSIDE	■		253
423	MEAS. RECTAN. INSIDE	■		258



Cycle number	Cycle name	DEF active	CALL active	Page
424	MEAS. RECTAN. OUTS.	■		263
425	MEASURE INSIDE WIDTH	■		267
426	MEASURE RIDGE WIDTH	■		271
427	MEASURE COORDINATE	■		275
430	MEAS. BOLT HOLE CIRC	■		280
431	MEASURE PLANE	■		285
441	FAST PROBING	■		300
460	CALIBRATION OF TS ON A SPHERE	■		318
461	TS CALIBRATION OF TOOL LENGTH	■		310
462	CALIBRATION OF A TS IN A RING	■		312
463	TS CALIBRATION ON STUD	■		315
480	CALIBRATE TT	■		333
481	CAL. TOOL LENGTH	■		336
482	CAL. TOOL RADIUS	■		340
483	MEASURE TOOL	■		345
484	CALIBRATE IR TT	■		350
620	VT 121 CALIBRATION	■		Further information: <b>VTC User's Manual</b> ID: 1322445-xx
621	MANUAL INSPECTION	■		Further information: <b>VTC User's Manual</b> ID: 1322445-xx
622	IMAGES	■		Further information: <b>VTC User's Manual</b> ID: 1322445-xx
623	BREAKAGE CHECK	■		Further information: <b>VTC User's Manual</b> ID: 1322445-xx
624	MEASUREMENT OF CUTTING EDGE ANGLE	■		Further information: <b>VTC User's Manual</b> ID: 1322445-xx
1400	POSITION PROBING	■		137
1401	CIRCLE PROBING	■		141
1402	SPHERE PROBING	■		146
1404	PROBE SLOT/RIDGE	■		151
1410	PROBING ON EDGE	■		73
1411	PROBING TWO CIRCLES	■		81
1412	INCLINED EDGE PROBING	■		89
1416	INTERSECTION PROBING	■		97
1420	PROBING IN PLANE	■		66

Cycle number	Cycle name	DEF active	CALL active	Page
1430	PROBE POSITION OF UNDERCUT	■		155
1434	PROBE SLOT/RIDGE UNDERCUT	■		161
1493	EXTRUSION PROBING	■		303
<b>Machining cycles</b>				
Cycle number	Cycle name	DEF active	CALL active	Page
13	ORIENTATION	■		357

**Index**

**3**

3D Touch Probes..... 40

**A**

About this manual..... 18  
 Automatic preset setting  
     Bolt hole circle..... 204  
     Center of 4 holes..... 214  
     Circle..... 141  
     Circular pocket (hole)..... 180  
     Circular stud..... 186  
     Fundamentals of 14xx..... 136  
     Fundamentals of 4xx..... 167  
     Inside corner..... 198  
     Outside corner..... 192  
     Rectangular pocket..... 169  
     Rectangular stud..... 174  
     Reference plane..... 241  
     Ridge..... 151  
     Ridge center..... 228  
     Ridge undercut..... 161  
     Single axis..... 219  
     Single position..... 137  
     Slot..... 151  
     Slot center..... 223  
     Slot undercut..... 161  
     Sphere..... 146  
     Touch probe axis..... 210  
     Undercut position..... 155

**B**

Basic rotation..... 107  
     Over two holes..... 111  
     over two studs..... 116, 121  
     Setting directly..... 130

**C**

Calibration  
     L probe..... 318  
     Normal probe..... 318  
 Calibration cycles..... 308  
     TS calibration in ring..... 312  
     TS calibration on sphere..... 318  
     TS calibration on stud..... 315  
     TS length..... 310  
 Calibration of tool touch probe  
     Calibration of IR TT..... 350  
     Calibration of TT..... 333  
 Classification of results..... 173

**D**

Determine inclined workpiece position  
     Touch probe cycles 14xx  
         fundamentals..... 55  
 Determining workpiece

misalignment  
     Basic rotation..... 107  
     Basic rotation over two holes 111  
     Basic rotation over two studs..... 116, 121  
     Fundamentals of touch probe cycles 400-405..... 106  
     Inclined edge probing..... 89  
     Intersection probing..... 97  
     Probing in plane..... 66  
     Probing on edge..... 73  
     Probing two circles..... 81  
     Rotation around C axis..... 126  
     Setting the basic rotation..... 130

**E**

Extrusion probing..... 303

**F**

Fast probing..... 300  
 Feature content level..... 22

**G**

GLOBAL DEF..... 48

**I**

Inclined workpiece position checking  
     Fundamentals..... 236

**M**

Measuring  
     Angle..... 245  
     Bolt hole circle..... 280  
     Circle outside..... 253  
     Coordinate..... 275  
     Hole..... 248  
     Inside width..... 267  
     Plane..... 285  
     Rectangle inside..... 258  
     Rectangle outside..... 263  
     Ridge width..... 271  
 Measuring in 3D..... 298  
 Measuring with Cycle 3..... 295

**O**

Option..... 21

**P**

Positioning logic..... 47  
 Probing feed rate..... 45

**R**

Recording measurement results..... 237

**S**

Software option..... 21  
 Spindle orientation..... 357

**T**

Table of cycles..... 360  
     Touch probe cycles..... 360  
 Tolerance monitoring..... 239  
 Tool compensation..... 240  
 Tool measurement  
     Complete measurement..... 345  
     Fundamentals..... 326  
     Machine parameters..... 329  
     Tool length..... 336  
     Tool radius..... 340  
     Tool table..... 331  
 Tool monitoring..... 239  
 Touch probe cycles 14xx  
     Circle probing..... 141  
     Evaluation of tolerances..... 62  
     Fundamentals..... 55  
     Inclined edge probing..... 89  
     Intersection probing..... 97  
     Position probing..... 137  
     Probing in plane..... 66  
     Probing on edge..... 73  
     Probing two circles..... 81  
     Ridge probing..... 151  
     Ridge undercut probing..... 161  
     Semi-automatic mode..... 57  
     Slot probing..... 151  
     Slot undercut probing..... 161  
     Sphere probing..... 146  
     Transfer an actual position..... 65  
     Undercut position probing..... 155

**W**

Workpiece, checking automatically  
     Polar preset..... 243

# 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](http://www.heidenhain.com/products/touch-probes-and-vision-systems)

